الجمهورية الجزائرية الديمقراطية الشعبية

**People's Democratic Republic of Algeria**

وزارة التعليم العالي والبحث العلمي

**Ministry of Higher Education and Scientific Research**

University of KasdiMerbah Ouargla
Faculty of Applied Sciences
Department: Mechanical Engineering

جامعة قاصدي مرباح ورقلة
كلية العلوم التطبيقية
قسم الهندسة الميكانيكية




***Presented in partial fulfillment of the requirements for the Master's Degree in the domain of Science and Technology***

***Department: Mechanical engineering***

***Specialty: Energy***

# Numericalstudy of air conditioningcharacteristics in offices for the evaluation of thermal comfort in the Saharan zone.

➢ **Presented by:**

DJEMIL Abdeldjalil

BELOUETTAR Issam

➢ **Supervisor:**

Dr. BOULEBBINA Cherif

**Academic Year: 2025/2026**

# Dedication

I dedicate this work to my beloved parents, whose unwavering love, constant encouragement, countless sacrifices, and heartfelt prayers have been the cornerstone of my achievements. Their support has been an endless source of strength and inspiration, and this accomplishment is a reflection of their dedication as much as my own.

To my family and friends, I extend my sincere gratitude for their continuous encouragement, understanding, and moral support throughout the most demanding stages of this academic journey.

To my respected classmate, Issam, and his family, I express my heartfelt appreciation for their friendship, kindness, generosity, and unwavering support.

I would also like to convey my profound gratitude to the faculty and staff of KasdiMerbah University of Ouargla for their guidance, patience, and academic assistance throughout the preparation of this master's thesis. Their expertise, valuable advice, and mentorship have contributed significantly to both my academic growth and professional development.

Finally, I am grateful to everyone who, directly or indirectly, contributed to the successful completion of this work.

**DjemilABDELdjalil**

# Dedication

With profound gratitude and appreciation, I dedicate this work to my dear parents, whose unwavering faith in me, unconditional love, and endless sacrifices have guided me throughout every stage of my life. Their encouragement, patience, and prayers have been a constant source of strength and determination, making this achievement possible.

To my family, thank you for your understanding, support, and confidence in my abilities.

To my friends and colleagues, I am deeply grateful for the memorable moments, valuable discussions, and continuous encouragement that accompanied me throughout this academic journey.

A special acknowledgment goes to my respected classmate, Djemil, and his family, whose kindness, generosity, friendship, and support have been sincerely appreciated and will always be remembered.

I would also like to express my sincere appreciation to the faculty members and staff of KasdiMerbah University of Ouargla for providing a stimulating academic environment and for their guidance and support during the completion of this master's thesis.

Finally, I dedicate this work to all those who contributed, directly or indirectly, to my education and personal growth.

**Issam BELOUETTAR**

# Acknowledgments

First and foremost, we express our deepest gratitude to Almighty God for granting us the health, perseverance, and strength required to accomplish this work and successfully complete this stage of our academic journey.

We are profoundly grateful to our families for their unwavering support, encouragement, patience, and sacrifices. Their constant belief in our abilities has been a source of motivation and inspiration throughout our studies.

We would like to convey our sincere appreciation to our supervisorDr.BOULEBBINACherif, whose expertise, valuable guidance, constructive advice, and continuous support were instrumental in the successful completion of this thesis. His mentorship greatly enriched both the quality of this work and our academic development.

We would also like to thank the distinguished members of the examination committee for accepting to evaluate this thesis and for dedicating their valuable time, knowledge, and expertise to its assessment.

Finally, we extend our heartfelt appreciation to all those who contributed, directly or indirectly, to the completion of this research. Their support, assistance, and encouragement have played an essential role in the success of this endeavor.

# Abstract

This study examines the performance of office air conditioning systems in desert regions usingCFD. Providing thermal comfort in hot climates is both crucial and challenging, requiring effective airflow strategies. The study analyzes heat transfer, airflow, and comfort indices such as PMV and PPD, according to international thermal comfort standards under the hot and dry climatic conditions of Ouargla. In this work, we compare two types of ventilation outlets with different wing configurations for an indoor cooling unit: one with a simple outlet (simple wing) and the other with a distributed outlet (Enhanced wing).

The results show that the simple wing achieves localized cooling and even heat distribution, with the lowest room temperature reaching approximately 27°C and the comfort indices exceeding acceptable limits (PPD >12%).

In contrast, the optimized wing achieved more uniform cooling, reducing the minimum room temperature to 23°C, lowering the indoor temperature by approximately 5°C, and improving comfort indicators (PPD ≈ 5%).

Furthermore, the results showed that the average airflow velocity when using the simple outlet (simple wing) was 0.2 m/s. In contrast, the distributed outlet (enhanced wing) recorded an average airflow velocity of 0.6 m/s.

Overall, the numerical study demonstrated the predictive effectiveness of cooling system behavior in desert climates, confirming that the indoor unit's airflow outlet plays a crucial role in optimizing both thermal comfort and energy efficiency.

# مـلـخـص

تتناول هذه الدراسة أداء أنظمة تكييف الهواء بالمكاتب في المناطق الصحراوية، باستخدام ديناميكا الموائع الحسابية (CFD).

يُعدّ توفير الراحة الحرارية في المناخات الحارة أمرً ابالغ الأهمية،ويُمثّل تحديًا في نفس الوقت، إذ يتطلب استراتيجيات فعّالة لتدفق الهواء. تُحلّل الدراسة انتقال الحرارة، تدفق الهواء، ومؤشرات الراحة مثل مؤشر PMV ومؤشر PPD حسب المعايير الدولية للراحة الحرارية تحت الظروف المناخية الحارة والجافة لمدينة ورقلة. في هذا العمل ،نقارن بين نوعين من منافذ التهوية ذات شكلين مختلفين لجناح وحدة التبريد الداخلية، الأول بمنفذ بسيط (الجناح البسيط) والثاني بمنفذ موزع (الجناح المُحسّن).

تُظهر النتائج أن الجناح البسيط يُحقق تبريدًا موضعيًا وتوزيعًا حراريًا متساويًا، حيث بلغت أدنى درجة حرارة الغرفة حوالي 27 درجة مئوية، وتجاوزت مؤشرات الراحة الحدود المقبولة (PPD>12%).

في المقابل، حقق الجناح المُحسّن تبريدًا أكثر تجانسًا، حيث خفّض أدنى درجة حرارة الغرفة إلى 23 درجة مئوية، وخفض درجة الحرارة الداخلية للهواء بحوالي 5 درجات مئوية، وحسّن مؤشرات الراحة(PPD≈ 5 %).علاوة على ذلك، أظهرت النتائج أن متوسط سرعة تدفق الهواء عند استخدام المنفذ البسيط (الحالة البسيطة)هو0.2م/ث.في المقابل، سجل المنفذ الموزع (الحالة المُحسّنة) متوسط سرعة تدفق هواء قدره0.6م/ث. بشكل عام، قدمت الدراسة العددية فعالية التنبؤ بسلوك نظام التبريد في المناخات الصحراوية،والتي تؤكد أن مخرج تدفق الهواء للوحدة الداخلية يلعب دورًا حاسمًا في تحسين كل من الراحة الحرارية وكفاءة الطاقة.

# *Table of Contents*

Dedication .....I
Dedication .....II
Acknowledgments.....III
Table of Contents .....VI
List of Figures.....VIII
List of Tables .....IX
Abbreviations and notations .....1
General Introduction .....1
Chapter I: Related Work .....1
I.1Introduction .....2
I.2Computational Fluid Dynamics (CFD): .....3
I.3 Using CFD in Air Conditioning System Analysis: .....4
I.4 Previous Studies on Air Conditioning System Analysis Using CFD: .....7
I.5 Studies Results: .....11
I.5.1 Study 1: .....11
I.5.3 Study 2: .....12
I.5.3 Study 3: .....14
I.5.4 Study 4: .....15
I.6Literature Review Summary: .....17
I.7Clarifying the Relationship between the Thesis Topic and Previous Studies:.....18
I.8Formulating the Theoretical Framework of the Problem:.....19
I.9Presenting the Scientific Hypotheses Underlying the Research: .....20
I.10Conclusion:.....21
Chapter II: Methodology of mathematical and numerical formulation .....24
II.1 Basic Equations:.....25
II.1.6 Turbulence Model:.....27
II.2 Thermal Comfort: .....27
II.3Control Domain (Geometry):.....29
II.4Boundary Conditions and Materials: .....31
II.5Mesh study:.....31
II.5.1 Mesh convergence: .....32
II.5.2Solution convergence:.....33
II.6Numerical solution validation:.....34
II.6.1 Comparison with experimental results: .....34
II.6.2 Physical plausibility:.....34
II.6.3 Mesh independence:.....34
II.6.4 Boundary condition consistency:.....34

II.7. Conclusion .....35
Chapter III: Results and Discussions .....36
III.1Introduction .....37
III.2 Validation of the Numerical Model: .....38
III.3Case 1: Simple wing: .....38
III.4Case 2: Enhanced wing .....42
III.5Comparative Discussion: .....45
III.6 Conclusion: .....46
References .....49

## *List of Figures*

Figure I.1: Air conditioning rooms design in Ansys Fluent with various duct positions (Study1).....7
Figure I.2: Air conditioning rooms design in Ansys Fluent with various duct positions (Study1).....8
Figure I.3: full-scale CFD model of the laboratory room with split AC unit (Study 3).....9
Figure I.4 : CFD model of cooled room with ACH variations.....10
Figure I.5:Room 1 Designed in ANSYS And Figure I.6: Temperature variations in Room 1 .....11
Figure I.7: Room 2 Designed in ANSYSFigure I.8:Temperature variations in Room 2 .....12
Figure I.9 Comparison of Temperature b/w All Room .....12
**Figure I.10**Case 1: Air-conditioner at z-axis wall .....13
**Figure I.11:** Temperature contour results for Case 1 .....13
**Figure I.12:** Air-conditioner at x-axis Case 2.....13
**Figure I.13:** Temperature contour results for Case 2 .....13
Figure I.14 : Comparison of simulated and experimental temperature distribution.....14
Figure I.15: Velocity distribution at different ACH values.....16
Figure I.16: Temperature contours at wrist and shoulder levels .....16

# *List of Tables*

TableI.1: Dimension Parameters (Study 1) .....7
Table I.2: Dimension Parameters (Study 2) .....8
Table I.3 : Dimension Parameters (Study 3) .....9
Table I.4:Dimension Parameters (Study 4). .....10
Table I.5 : Comparison of Results with all three cases of AC Room.....12
Table I.6: Comparison of Results with all three cases of HVAC unit placement .....14
Table I.7: Comparison of Simulation and Experimental Results .....14
Table I.8: Comparison of Comfort Results Across ACH Values.....16
Table I.9Comparative results of study cases .....45

## *Abbreviations and notations*

### ➢ Abbreviations

**CFD**Computational Fluid Dynamics

**HVAC**Heating, Ventilation, and Air Conditioning

**PMV**Predicted Mean Vote

**PPD**Predicted Percentage of Dissatisfied

**ASHRAE** American Society of Heating, Refrigerating and Air-Conditioning Engineers

**ISO**International Organization for Standardization

**AC**Air Conditioning

**UDF**User Defined Function (in ANSYS Fluent)

**RANS**Reynolds-Averaged Navier–Stokes

**LES**Large Eddy Simulation

### ➢ Notations

$\boldsymbol{T}$ Temperature ($K$ *or* $°C$)

$\boldsymbol{V}$ Velocity ($m/s$)

$\boldsymbol{P}$ Pressure ($Pa$)

$\boldsymbol{\rho}$ Density ($kg/m^3$)

$\boldsymbol{\mu}$ Dynamic viscosity ($Pa{\cdot}s$)

$\boldsymbol{k}$ Turbulent kinetic energy ($m^2/s^2$)

$\boldsymbol{\varepsilon}$ Turbulent dissipation rate ($m^2/s^3$)

$\boldsymbol{q}$ Heat flux ($W/m^2$)

$\dot{\mathbf{m}}$ Mass flow rate (kg/s)

$\boldsymbol{\phi}$ Generic scalar variable (temperature, concentration, etc.)

$\boldsymbol{\nabla}$ Gradient operator

$\boldsymbol{\Delta T}$ Temperature difference (K or °C)

$\boldsymbol{\theta}$ Thermal comfort index (dimensionless)

# General Introduction

Thermal comfort in Saharan office environments represents a major engineering challenge, as outdoor temperatures frequently exceed 50 °C. In such hyper-arid climates, air-conditioning systems are indispensable for maintaining indoor microclimates that safeguard occupant well-being and productivity. Comfort assessment is based on standardized indices such as PMV (Predicted Mean Vote) and PPD (Predicted Percentage of Dissatisfied), as defined by ASHRAE Standards.

In this context, several strategies have been investigated to improve thermal comfort while reducing the energy consumption associated with air-conditioning systems. One such approach involves integrating ceiling fans with higher air-conditioning setpoints. Field measurements and questionnaire surveys have been conducted to assess the resulting thermal comfort conditions and energy-saving potential under different operating conditions. These findings highlight the potential of combining air movement with optimized cooling setpoints as an energy-efficient strategy for maintaining acceptable thermal comfort in office environments.

Building on these approaches to improve both thermal comfort and HVAC energy efficiency, this study employs Computational Fluid Dynamics (CFD) to evaluate the performance of HVAC systems in Saharan offices. The first chapter reviews airflow dynamics, heat transfer processes, and thermal comfort indices, highlighting the importance of efficient conditioning design. The second chapter presents the mathematical and numerical formulation, including governing equations, turbulence modeling, boundary conditions, and mesh generation. The computational domain incorporates realistic office geometry with walls, furniture, and occupants modeled as thermal sources. Convergence and validation against experimental data under Saharan climatic conditions ensure methodological reliability.

The third chapter compares two diffuser configurations:

- Case I: conventional wing-type outlet.
- Case II: enhanced wing with distributor fins.

Results demonstrate that the enhanced configuration improves airflow distribution, reduces thermal stratification, and achieves better compliance with PMV/PPD indices.

The study concludes by synthesizing findings on airflow, indoor temperature, and comfort indicators, offering recommendations for climate-adaptive HVAC design. Overall, the work establishes a rigorous experimental–numerical framework and provides practical insights into conditioning strategies tailored for Saharan offices, balancing thermal comfort with energy efficiency.

# *Chapter I: Airflow and Heat Transfer Principles*

## I.1Introduction:

Air conditioning systems are among the most important technologies integrated into modern buildings to ensure an acceptable indoor environment. Their role is particularly prominent in hot, dry regions, especially desert climates, where ambient temperatures often exceed 45-50°C. In such environments, inadequate cooling directly impacts human comfort, health, and productivity in the workplace, especially in offices where occupants spend long periods under heat stress.

Achieving thermal comfort within buildings depends on several interconnected factors, most notably air distribution, temperature gradients, humidity levels, and airflow velocity within the occupied space. Therefore, the design of heating, ventilation, and air conditioning (HVAC) systems must be scientifically precise to ensure uniform air distribution and minimize thermal variations. Furthermore, thermal comfort is quantified using internationally recognized indicators such as the Average Expected Vote (PMV) Index and the Expected Percentage Dissatisfaction Index (PPD), according to the standards of the American Society of Heating, Refrigerating and Air Conditioning Engineers (ASHRAE). These indicators link physical parameters (temperature, velocity, humidity, metabolic rate, and insulation of clothing) to human perception, ensuring that system design aligns with physiological and psychological comfort requirements.

With the rapid development of numerical simulation techniques, computational fluid dynamics (CFD) has emerged as an effective tool for analyzing airflow behavior and heat transfer in conditioned spaces. CFD enables a detailed visualization of velocity fields, temperature lines, and turbulence effects, providing predictive insights into system performance. Beyond visualization, CFD facilitates the optimization of diffuser placement, inlet configurations, and cooling strategies, reducing reliance on costly experimental campaigns. Furthermore, CFD-based studies provide a robust methodological framework for validating HVAC system designs against experimental data and international standards, ensuring reliability under the harsh climatic conditions characteristic of desert regions.

## I.2Computational Fluid Dynamics (CFD):

Computational Fluid Dynamics (CFD) is a discipline within computational mechanics that focuses on predicting fluid flow and heat transfer phenomena through numerical modeling. It is based on the discretization and solution of the fundamental conservation laws governing fluid motion:

Continuity Equation → ensures mass conservation, stating that the mass entering a control volume must equal the mass leaving it.

Navier–Stokes Equations → represent momentum conservation, derived from Newton's second law applied to viscous fluids, incorporating pressure forces, viscous stresses, and external body forces.

Energy Equation → expresses energy conservation, accounting for heat transfer, work interactions, and energy transport within the flow.

By solving these equations, CFD provides detailed insight into the distribution of velocity, pressure, and temperature fields in complex systems such as air-conditioned indoor environments.

Specialized solvers, including ANSYS Fluent, are commonly employed to perform these calculations. The computational domain is subdivided into a finite number of small elements (mesh), and the governing equations are solved iteratively within each element. This process yields a high-resolution numerical representation of fluid and thermal behavior, enabling engineers to evaluate system performance, optimize design parameters, and validate results against experimental data [2].

## I.3 Using CFD in Air Conditioning System Analysis:

CFD definition: Computational Fluid Dynamics (CFD) is a numerical technique that employs mathematical models and algorithms to simulate fluid flow, heat transfer, and related physical phenomena.

Examples of CFD applications:

HVAC and Building Design: CFD is widely used to optimize heating, ventilation, and air conditioning systems, ensuring uniform air distribution, improved thermal comfort, and reduced energy consumption in residential, commercial, and industrial buildings.

Automotive and Aerospace Engineering: Engineers employ CFD to study aerodynamics, reduce drag, enhance fuel efficiency, and design efficient cooling systems for engines, brake disks, and exhaust components. In aerospace, CFD supports wing design, cabin airflow, and propulsion system analysis.

Pipeline and Industrial Safety: CFD simulations help predict fluid flow in pipelines, detect risks of leakage or blockage, and evaluate the impact of defects or repairs. This contributes to safer operation and better maintenance strategies in oil, gas, and water distribution networks In addition, CFD is increasingly applied to multiphase flows, combustion modeling, and pollutant dispersion studies, providing predictive insights into environmental safety and industrial risk management.

In recent years, Computational Fluid Dynamics (CFD) has become a powerful tool for analyzing and optimizing the performance of air conditioning and ventilation systems in buildings. By numerically solving the governing equations of fluid flow and heat transfer, CFD enables researchers and engineers to visualize and predict how conditioned air behaves within enclosed spaces. These simulations provide valuable insights into airflow patterns, temperature distribution, and the interaction between ventilation openings, ductwork, and occupants.

CFD studies are particularly useful in evaluating the distribution of cool air within rooms, identifying regions of thermal stratification, and detecting potential zones of discomfort. For instance, improper duct placement may lead to stagnant air pockets or uneven cooling, while optimized duct layouts can ensure uniform temperature fields and improved occupant comfort. Beyond comfort, CFD also supports the design of energy-efficient HVAC systems, helping reduce cooling loads and electricity consumption by minimizing recirculation losses and avoiding overcooling Moreover, CFD allows sensitivity analyses of different boundary conditions, turbulence models, and diffuser geometries, enabling engineers

to refine system design and validate performance against ASHRAE standards and experimental benchmarks.

## I.4 Key Definitions and Concepts:

### I.4.1Thermal Comfort:

**Definition:**Thermal comfort is the condition of mind that expresses satisfaction with the thermal environment.

**Determinants:**Air temperature, humidity, air velocity, and radiant heat exchange with surrounding surfaces.

**Indices:**

- **PMV (Predicted Mean Vote):**A statistical index predicting the average thermal sensation of a group of occupants It integrates environmental parameters (temperature, humidity, air velocity, mean radiant temperature) with personal factors (metabolic rate, clothing insulation), providing a quantitative measure of perceived comfort.
- **PPD (Predicted Percentage of Dissatisfied):**Indicates the percentage of people likely to feel uncomfortable It is mathematically linked to PMV and reflects the expected proportion of dissatisfied occupants, even under optimal conditions, acknowledging that perfect comfort cannot be achieved for all individuals.

**Acceptable Ranges:**Acceptable Ranges: For good comfort, PMV should lie between −0.5 and +0.5, while PPD should remain below 10% These thresholds are defined by ASHRAE Standard 55 and ISO 7730, serving as benchmarks for HVAC system design and validation.

CFD simulations allow engineers to map PMV and PPD values across a room, highlighting areas where occupants may feel too warm or too cold. This spatial analysis is crucial for validating HVAC designs before implementation Such mapping enables sensitivity studies of diffuser placement, duct geometry, and ventilation rates, ensuring that comfort indices remain within acceptable ranges across diverse occupancy scenarios.

### I.4.2HVAC System Design:

**Definition:** HVAC (Heating, Ventilation, and Air Conditioning) systems regulate indoor climate by controlling temperature, humidity, and air quality.

**Design Considerations:**

- Duct Locations: Placement of supply and return ducts strongly influences airflow distribution.
- Airflow Rates: Must balance between sufficient cooling and energy efficiency.
- Outlet Temperatures: Should be optimized to avoid drafts or thermal discomfort.

CFD enables virtual prototyping of HVAC layouts, allowing engineers to test different duct configurations, diffuser types, and ventilation strategies without costly physical trials.

### I.4.3Energy Efficiency:

**Definition:** Energy efficiency in HVAC systems refers to minimizing cooling loads and electricity consumption while maintaining thermal comfort It involves optimizing system design to reduce recirculation losses, avoid overcooling, and integrate passive or renewable cooling strategies.

**Challenges:**

- Poor duct placement can cause uneven cooling, leading to higher energy demand.
- Oversized systems waste energy, while undersized systems fail to meet comfort requirements.

### I.4.4CFD Contribution:

- Identifies inefficiencies in airflow distribution.
- Suggests optimized layouts that reduce wasted energy.
- Supports integration of passive cooling strategies (e.g., natural ventilation, shading).

By combining CFD with building energy simulation tools, engineers can evaluate both thermal comfort and energy performance, ensuring sustainable HVAC designs.

### I.4.5Broader Applications of CFD in HVAC:

**Indoor Air Quality:** CFD can track pollutant dispersion, $CO_2$ concentration, and ventilation effectiveness.

**Occupant Safety:** Used to model smoke movement in case of fire, aiding in evacuation planning.

**Climate-Specific Design:** In hot climates such as the Saharan region, CFD helps design systems that cope with extreme solar loads and high ambient temperatures.

**Integration with Renewable Energy:** CFD supports hybrid cooling systems that combine mechanical ventilation with solar-assisted or evaporative cooling technologies.

## I.5 Previous Studies on Air Conditioning System Analysis Using CFD:

Patel and Dhakar (2018) conducted a numerical investigation to evaluate the performance of an air-conditioning system in a confined room using ANSYS Fluent. Their primary objective was to assess air distribution and temperature variations as a function of duct location. Three duct layouts were tested, and results demonstrated that duct positioning exerts a significant influence on cooling efficiency and occupant comfort Specifically, the study highlighted that improper duct placement can lead to thermal stratification, stagnant air pockets, and localized discomfort, whereas optimized duct configurations promote uniform temperature fields and improved energy efficiency.

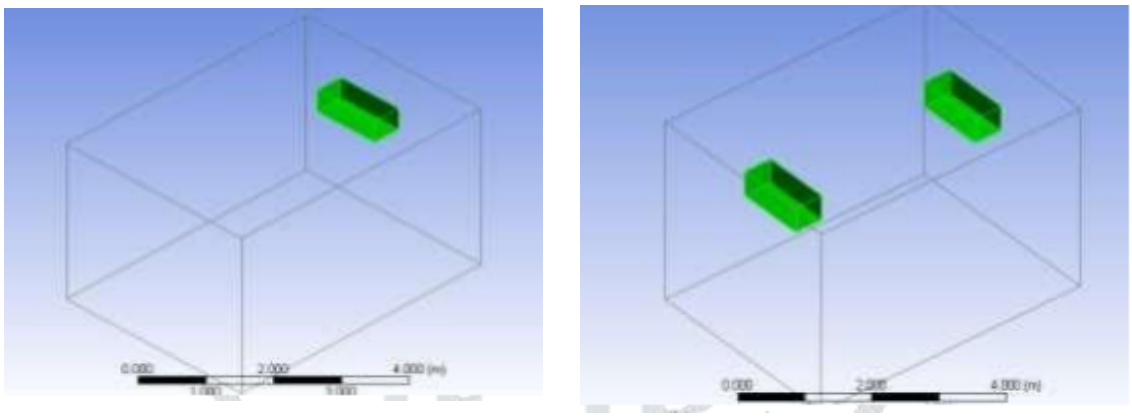
**Three duct layouts were investigated:**

- **Case 1:** A single air duct positioned approximately 9 feet above the floor.
- **Case 2:** Two air ducts placed opposite each other, maintaining the same mass flow rate.

The dimensions of the room used in the study were approximately:

**TableI.1:** Dimension Parameters (Study 1)

| Parameters | Room dimensions (meter) |
|---|---|
| Length: | 4 |
| Width: | 3.65 |
| Height: | 6 |



Case 1 Case 2

**Figure I.1:** Air conditioning rooms design in Ansys Fluent with various duct positions (Study1)

The temperature of the air entering through the air-conditioning ducts was approximately 293 K (20 °C), while the temperature of the room walls was maintained at around 40 °C to replicate high-temperature boundary conditions.

In addition, Barau et al. (2024) carried out a study titled "Observation on Air Flow Distribution in Room by HVAC System using ANSYS Fluent." This research analyzed two different air-conditioner placements — wall-mounted along the z-axis and wall-mounted along the x-axis — under varying inlet velocities. The study highlighted the importance of strategic air unit placement, showing that optimized configurations improve airflow distribution, enhance thermal comfort, and reduce energy consumption The authors emphasized that ceiling-mounted configurations, though not the focus of this comparison, generally outperform wall-mounted units in minimizing thermal stratification. Their findings also demonstrated that higher inlet velocities improve mixing but may increase local draft discomfort, underscoring the need for balanced design strategies.

Two duct layouts were investigated:

- Case 1: Air-conditioner model at the center along the z-axis on the wall.
- Case 2: Air-conditioner model at the center along the x-axis on the wall.

The dimensions of the room used in the study were approximately: The computational domain was discretized into a structured mesh, and boundary conditions were applied to replicate realistic indoor environments. Validation against experimental data confirmed the reliability of the numerical approach, making the study a valuable reference for HVAC design in hot-arid climates.

**Table I.2:** Dimension Parameters (Study 2)

| Parameters | Room dimensions (meter) |
|---|---|
| Length: | 8 |
| Width: | 7 |
| Height: | 4 |



Case 1 Case 2

**Figure I.2:** Air conditioning rooms design in Ansys Fluent with various duct positions (Study1)

Moreover, Zhu et al. (2023) conducted a full-scale numerical simulation and experimental validation of the thermal environment in an air-conditioned room under heating mode. Their objective was to accurately predict airflow and temperature distribution by combining CFD with experimental measurements. Unlike simplified models, this study incorporated detailed components of the split AC unit such as the crossflow fan and heat exchangers, ensuring realistic simulation of indoor conditions This inclusion of internal AC components allowed the model to capture secondary flow structures, recirculation zones, and localized turbulence effects that are often neglected in simplified HVAC simulations.

The physical model was based on a real laboratory room with dimensions of approximately:

**Table I.3 :** Dimension Parameters (Study 3)

| Parameters | Room dimensions (meter) |
|---|---|
| Length: | 4.39 |
| Width: | 3.93 |
| Height: | 2.90 |

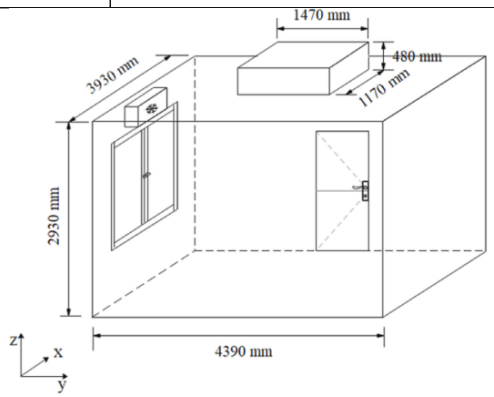



**Figure I.3:**full-scale CFD model of the laboratory room with split AC unit (Study 3).

The air-conditioner was installed between the ceiling and the top of the window, and the CFD model employed ~10 million mesh cells to capture airflow details. The Shear-Stress Transport (SST) k–ω turbulence model was applied to simulate both low-Reynolds flows near walls and high-Reynolds flows near fan blades the choice of SST k–ω was particularly suitable for this case, as it provides accurate near-wall treatment while maintaining robustness in free-stream regions, making it ideal for mixed convection and forced airflow scenarios.

To validate the simulation, 210 thermocouple sensors were distributed throughout the room. Results showed that 194 out of 210 points had deviations below ±10%, confirming the accuracy of CFD predictions This high level of agreement between numerical and experimental data underscores the reliability of CFD in replicating real indoor thermal

environments, and highlights the importance of full-scale validation for HVAC system design in extreme climates.

Additionally, Ozsagiroglu et al. (2021) carried out a CFD analysis of thermal comfort conditions in a cooled room. Their objective was to evaluate how different air exchange rates (ACH) influence indoor comfort levels according to ASHRAE-55 standards.

The study modeled a small closed room with human and computer heat sources, incorporating both convection and radiation heat transfer effects This dual consideration of convective and radiative exchanges allowed the model to capture more realistic thermal interactions, particularly the influence of radiant surfaces on occupant comfort.

The physical model was based on a rectangular room with dimensions of approximately:

**Table I.4:**Dimension Parameters (Study 4).

| Parameters | Room dimensions (meter) |
|---|---|
| Length: | 1.8 |
| Width: | 1.8 |
| Height: | 2.85 |

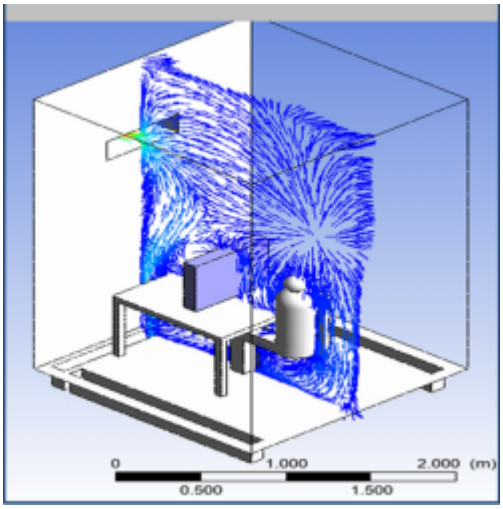



**Figure I.4 :** CFD model of cooled room with ACH variations

The CFD simulations were performed using ANSYS Fluent, with air supplied at 15 °C under varying ACH values (1, 3, 5, 10, 15). Comfort indices were evaluated at wrist level (0.1 m) and shoulder level (1.1 m), considering metabolic activity (1.2 met) and clothing insulation (0.67 clo) the multi-level evaluation highlighted vertical temperature gradients, showing that wrist-level measurements tended to be cooler than shoulder-level, an important factor in assessing occupant comfort across different body zones.

These studies collectively demonstrate that CFD simulations are powerful tools for evaluating HVAC system performance, providing insights into duct placement, airflow

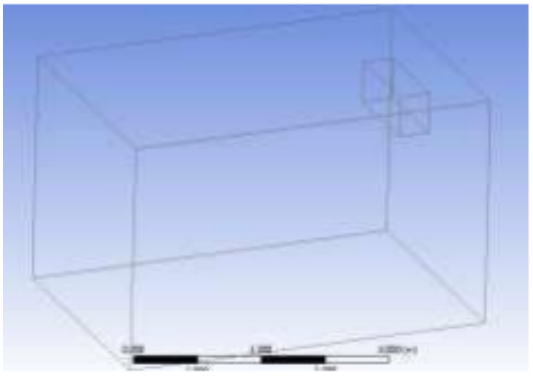
velocity, and energy efficiencymore importantly, they establish CFD as a validated methodology for predicting thermal comfort indices, identifying discomfort zones, and guiding the design of climate-adaptive HVAC systems in hot-arid regions.

## I.6Studies Results:

### I.6.1Study 1:

Numerical simulation resultsshowedthat the location of air ductssignificantly affects air distribution and temperaturewithin a room the studyconfirmedthatductpositioning influences airflowuniformity, thermal stratification, and coolingefficiency. Improper placement cancreate stagnant zones and uneventemperaturefields, whileoptimizedlayoutsenhancemixing, reduce hot spots, and improveoverall occupant comfort.

In the first scenario, where a single air ductwasused, the lowesttemperatureachievedinside Room 1 wasapproximately 36 °C.



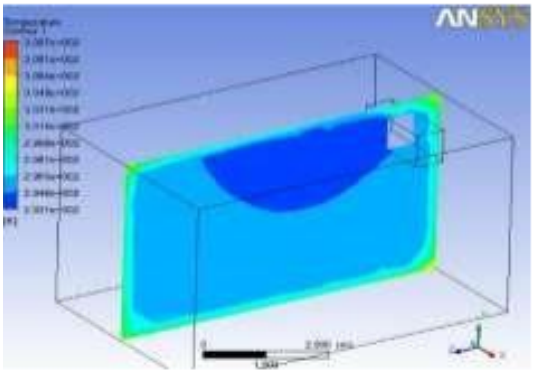


**Figure I.5:**Room 1 Designed in ANSYS**Figure I.6:** Temperature variations in Room 1

In the second scenario, where two ducts were used, air distribution within Room 2 improved, and the minimum temperature decreased to approximately 33 °C the improved configuration enhanced mixing efficiency, reduced vertical temperature gradients, and minimized stagnant air pockets compared to the single-duct case. This resulted in a more uniform thermal field, better compliance with comfort indices (PMV/PPD), and lower localized discomfort. The study also noted that balanced duct placement contributes to reduced cooling loads, thereby improving overall energy efficiency.

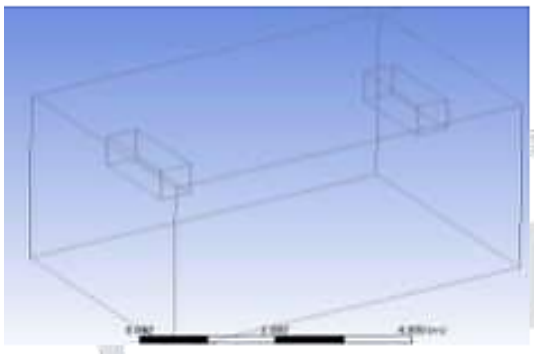
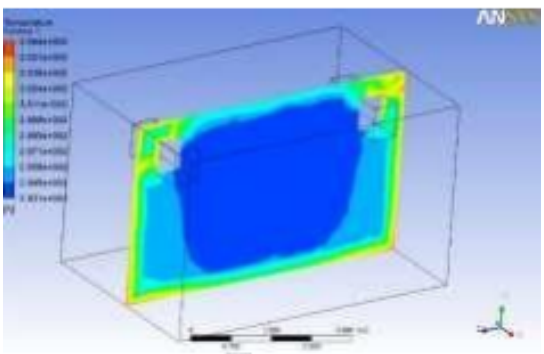


**Figure I.7**: Room 2 Designed in ANSYS



**FigureI.8:**Temperature variations in Room 2

**Table I.5 :** Comparison of Results with all three cases of AC Room

| Rooms | Minimum Temperature of Room (°C) |
|---|---|
| Room 1 | 36 |
| Room 2 | 33 |

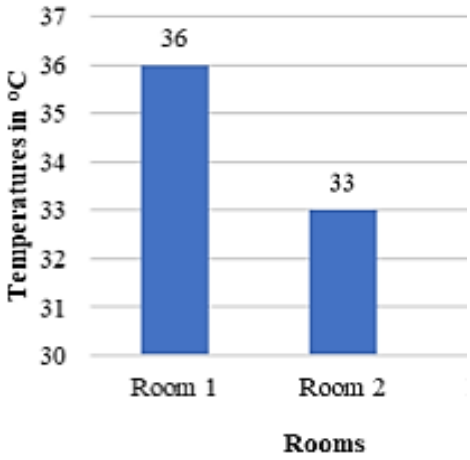



**Figure I.9:** Comparison of Temperature b/w All Room.

The study also demonstrated that using two air ducts in the room's ceiling allows for better air circulation, resulting in faster cooling and more even heat distribution.

### I.6.2Study 2:

Numerical simulation results showed that the position of the air-conditioning unit significantly affects airflow distribution, pressure, and temperature within the room.

In the first scenario (Case 1), where the unit was mounted on the wall along the z-axis, cooling improved with higher inlet velocities, but the average room temperature remained relatively high, around 294.6 K (21.4 °C) at 1.5 m/s, decreasing slightly to 293.2 K (20 °C) at 3.5 m/s the results indicated that while increasing inlet velocity enhances mixing and reduces localized hot zones, it does not fully eliminate thermal stratification. Moreover, higher

velocities may introduce draft discomfort near occupied zones, highlighting the trade-off between improved cooling and occupant comfort. The study emphasized that wall-mounted z-axis placement is less effective compared to ceiling-mounted configurations, which generally achieve more uniform temperature distribution and better compliance with thermal comfort indices.

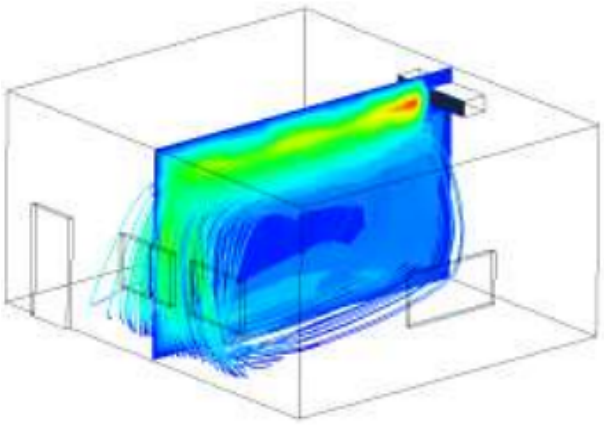


**Figure I.10**Case 1: Air-conditioner at z-axis wall

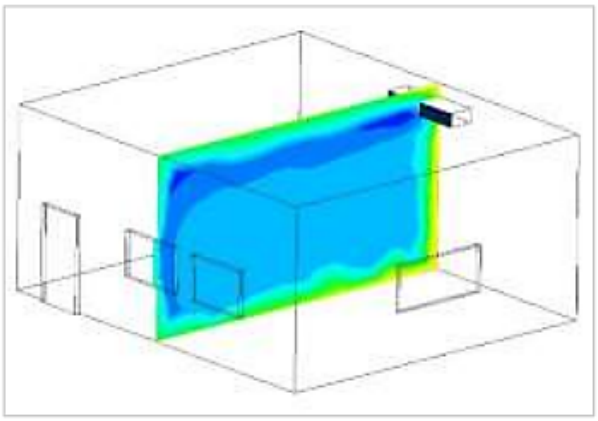


**Figure I.11:** Temperature contour results for Case 1

In the second scenario (Case 2), where the unit was mounted on the wall along the x-axis, airflow distribution improved compared to Case 1, but cooling was still less effective than the ceiling-mounted configuration. The lowest temperature achieved was 293.3 K (20.1 °C) at 3.5 m/s the improved x-axis placement enhanced lateral mixing and reduced localized hot zones compared to the z-axis case, yet vertical stratification persisted. Although this configuration provided better comfort indices than Case 1, it remained less efficient in achieving uniform cooling across the occupied zone. The study concluded that while wall-mounted x-axis units offer moderate improvements, ceiling-mounted systems remain superior in balancing airflow, minimizing drafts, and ensuring energy-efficient operation.

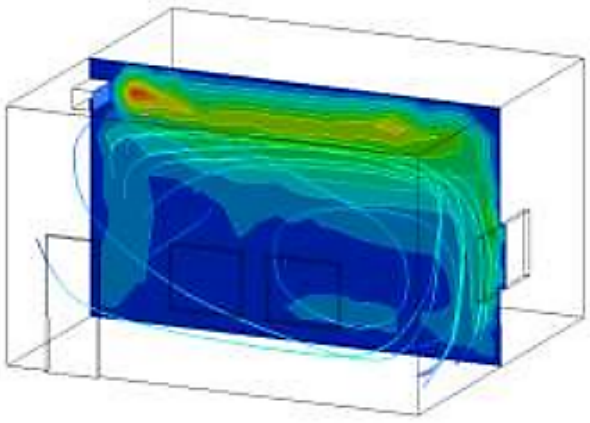


**Figure I.12:** Air-conditioner at x-axis Case 2

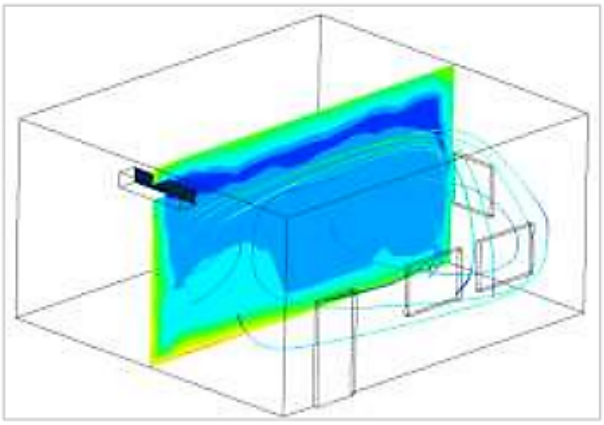


**Figure I.13:** Temperature contour results for Case 2

**Table I.6:** Comparison of Results with all three cases of HVAC unit placement

| Case | Minimum Temperature (°C) |
|---|---|
| Case 1 (Wall – z-axis) | ~21.4°C → 20°C |
| Case 2 (Wall – x-axis) | ~20.1°C |

The study demonstrated that wall-mounted configurations (z-axis and x-axis) influence airflow distribution and cooling efficiency, with the x-axis placement providing slightly better results.

### I.5.3 Study 3:

Numerical simulation results demonstrated that buoyancy effects in heating mode caused warmer air stratification near the ceiling, while cooler air remained at lower levels, with the simulated supply airflow rate reaching 537.48 m³/h and showing only a 4.97% deviation from experimental data; the temperature distribution revealed higher values in the upper region of the room due to the upward movement of supply air, and validation against experimental measurements confirmed CFD's reliability, as most data points showed less than 10% error this agreement between numerical and experimental results highlights the robustness of the CFD methodology, particularly when detailed AC components and advanced turbulence models are incorporated. The study further emphasized that buoyancy-driven stratification is a critical factor in heating mode, requiring careful diffuser placement and velocity control to avoid discomfort zones near the ceiling. Such validated simulations provide engineers with confidence in applying CFD to optimize HVAC designs under extreme climatic conditions.

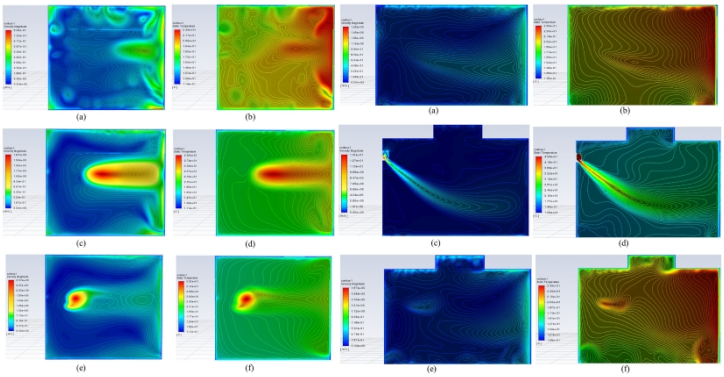


**Figure I.14 :** Comparison of simulated and experimental temperature distribution[….].

**Table I.7:** Comparison of Simulation and Experimental Results

| Parameter | Simulation | Experiment | Deviation |
|---|---|---|---|
| Supply airflow rate (m³/h) | 537.48 | 564.5 | 4.97% |
| Temperature deviation points | 194/210 < ±10% | _ | _ |

This study highlighted the importance of full-scale validation, showing that CFD can accurately predict indoor thermal environments when detailed AC components and buoyancy effects are considered The integration of experimental measurements with high-resolution CFD modeling demonstrated that numerical simulations can reliably capture complex thermal phenomena such as stratification, recirculation, and localized turbulence. Such validated approaches not only strengthen confidence in CFD as a predictive tool but also provide engineers with practical guidance for diffuser placement, airflow control, and energy optimization in real-world HVAC applications.

### I.5.4 Study 4:

Numerical simulation results showed that the ventilation rate strongly influences thermal comfort The study demonstrated that airflow rate directly impacts the balance between heat removal, occupant comfort, and draft risk. Insufficient ventilation leads to overheating and discomfort, while excessive ventilation can cause turbulence and localized drafts. Optimal ACH values ensure compliance with ASHRAE-55 standards, highlighting the importance of carefully selecting ventilation rates during HVAC design:

- At ACH = 1, the room remained overheated, failing to meet comfort standards.
- At ACH = 3–10, PMV and PPD values fell within acceptable ranges, providing optimal comfort.
- At ACH = 15, excessive airflow caused draft discomfort, reducing occupant satisfaction.
- Wrist-level measurements indicated cooler sensations compared to shoulder-level, confirming vertical temperature gradients.

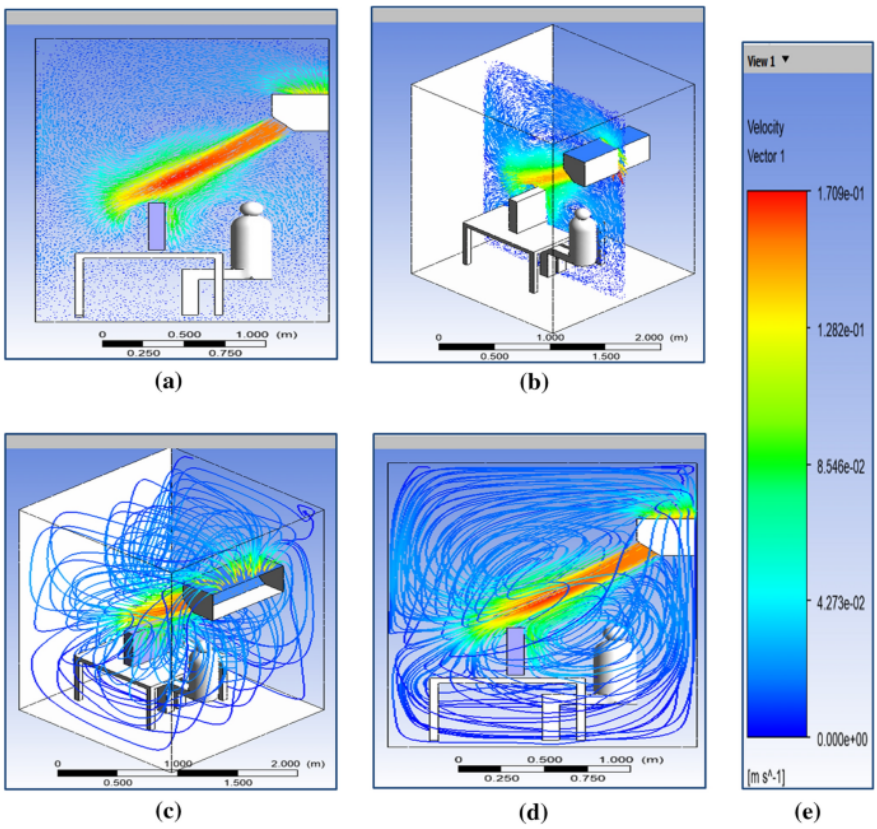



**Figure I.15:** Velocity distribution at different ACH values

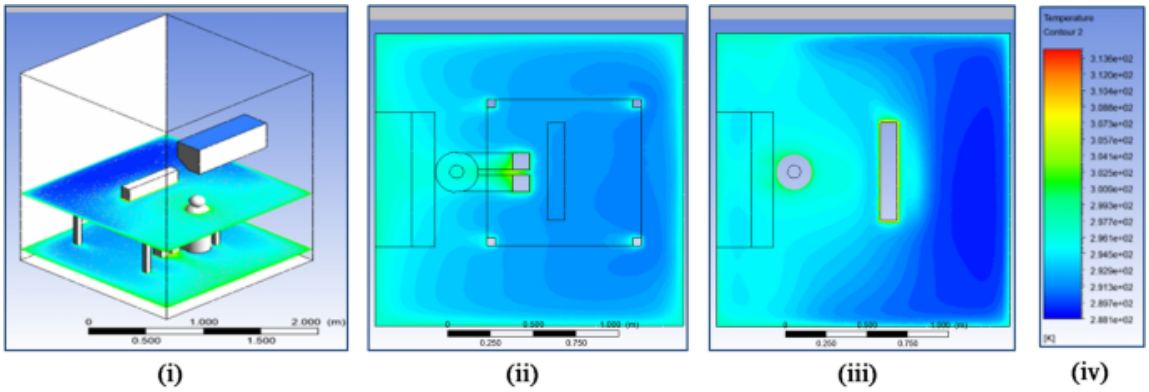



**Figure I.16:** Temperature contours at wrist and shoulder levels

**Table I.8:** Comparison of Comfort Results Across ACH Values

| ACH Value | Comfort Outcome | PMV/PPD Compliance |
|---|---|---|
| 1 | Overheated, poor comfort | Not acceptable |
| 3-10 | Optimal comfort | Within ASHRAE-55 |
| 15 | Draft discomfort | Not acceptable |

This study demonstrated that ventilation rate optimization is critical for achieving thermal comfort in cooled rooms, reinforcing the importance of balancing airflow to avoid both overheating and draft discomfort the findings underline that ventilation design must account not only for average room temperature but also for localized comfort indices such as PMV

and PPD. Excessively low ACH values lead to insufficient cooling and discomfort, while excessively high ACH values introduce turbulence and draft sensations. Optimal ventilation rates therefore represent a compromise between thermal uniformity, energy efficiency, and occupant satisfaction, serving as a key design parameter in HVAC engineering.

## I.6Literature Review Summary:

Previous research has consistently highlighted the importance of air-conditioning system design and the strategic placement of air ducts in achieving optimal indoor thermal comfort. Studies demonstrate that duct location, diffuser type, and airflow rate significantly influence the distribution of conditioned air within enclosed spaces. Poorly designed layouts often result in uneven cooling, thermal stratification, or zones of discomfort, whereas optimized duct configurations enhance both occupant satisfaction and system efficiency for example, comparative CFD studies have shown that dual-duct systems reduce peak room temperatures by up to 3 °C compared to single-duct layouts, while optimized diffuser placement minimizes stagnant zones and improves compliance with PMV/PPD indices.

In parallel, the advancement of numerical simulation techniques has provided researchers with powerful tools to investigate these phenomena. Among these, **Computational Fluid Dynamics (CFD)** has emerged as a particularly effective method for analyzing airflow and temperature distribution in air-conditioned environments. Using CFD, engineers can simulate complex interactions between air supply, return ducts, and indoor surfaces, thereby predicting how design choices impact both comfort and energy consumption the ability to incorporate turbulence models (e.g., k–ε, SST k–ω) and radiation effects allows CFD to capture both convective and radiative heat transfer, making it a comprehensive tool for HVAC analysis.

Software platforms such as **ANSYS Fluent** have been widely adopted in this field, offering robust solvers for fluid flow, heat transfer, and turbulence modeling. These tools allow for detailed visualization of airflow patterns, velocity fields, and temperature contours, enabling researchers to evaluate the performance of HVAC systems under varying operating conditions. The ability to conduct virtual experiments reduces reliance on costly physical prototypes and accelerates the design process furthermore, mesh refinement and grid independence studies ensure numerical accuracy, while validation against experimental data strengthens confidence in CFD predictions.

Several studies have demonstrated the effectiveness of CFD in identifying inefficiencies within HVAC systems and proposing corrective measures. For example, simulations have been used to assess the impact of diffuser placement on thermal comfort indices such as **PMV** and **PPD**, as well as to evaluate energy-saving strategies through optimized duct layouts. The

literature also emphasizes the role of CFD in supporting sustainable building design, particularly in climates where cooling loads are high and energy efficiency is critical in hot-arid regions, CFD has been applied to integrate passive cooling strategies such as natural ventilation, PCM-enhanced walls, and shading devices, demonstrating significant reductions in overheating and energy demand.

Building on this foundation, the present work aims to conduct a numerical study focused on offices located in desert regions, where extreme ambient temperatures and intense solar radiation pose unique challenges to indoor climate control. By applying CFD techniques, this study seeks to analyze air distribution and temperature fields within office spaces, with the goal of achieving optimal thermal comfort conditions while minimizing energy consumption. The outcomes are expected to provide practical insights into HVAC design strategies tailored for hot, arid climates, contributing to both occupant well-being and sustainable energy use ultimately, this research bridges the gap between theoretical CFD modeling and practical HVAC applications, offering validated design recommendations for Saharan office environments.

## I.7Clarifying the Relationship between the Thesis Topic and Previous Studies:

Previous studies, such as those conducted by Patel and colleagues (2018), have primarily focused on analyzing air distribution and temperature fields within air-conditioned rooms using Computational Fluid Dynamics (CFD) techniques implemented in ANSYS Fluent. These works compared the impact of different duct locations, diffuser types, and ventilation strategies on airflow uniformity and occupant comfort. Their findings confirmed that duct placement and system design are critical determinants of both thermal comfort indices (such as PMV and PPD) and overall energy efficiency in HVAC systems.

This line of research provides a strong methodological foundation for the present thesis. Our study shares a similar objective—evaluating air conditioning system performance and thermal comfort through numerical simulation—but introduces a distinct environmental context. Unlike the controlled indoor environments typically studied in temperate climates, our focus is on office buildings located in desert regions, where extreme ambient temperatures, high solar radiation, and low humidity present unique challenges. These conditions amplify the importance of duct location, airflow rate, and ventilation design, as improper configurations can lead to excessive cooling loads, discomfort, and unsustainable energy consumption.

By situating the analysis within a desert climate framework, this thesis extends the scope of previous CFD-based HVAC studies. It adapts the theoretical and technical approaches established in earlier research to a new environmental setting, requiring adjustments in boundary conditions, material properties, and air characteristics to ensure accurate simulation results. In doing so, the study not only validates the applicability of CFD in diverse climates but also contributes practical insights into designing energy-efficient cooling systems for arid regions.

Therefore, the relationship between this thesis and prior studies can be summarized as follows:

- **Methodological Continuity:** Building upon established CFD techniques and ANSYS Fluent workflows used in earlier HVAC research.
- **Contextual Innovation:** Applying these methods to desert office environments, where climatic extremes demand tailored design solutions.
- **Practical Contribution:** Offering design recommendations that balance thermal comfort with energy efficiency, specifically for regions where cooling demand is both critical and costly.

In essence, previous studies provide the theoretical and technical framework, while this thesis advances the field by applying CFD analysis to a new climatic context, thereby broadening the relevance and applicability of simulation-based HVAC design.

## I.8Formulating the Theoretical Framework of the Problem:

The theoretical framework of this study is constructed to address the unique challenge of cooling office spaces in desert climates, where extreme heat and aridity intensify the demand for efficient HVAC systems. Unlike general CFD applications in building ventilation, this framework specifically integrates three interrelated dimensions that guide the research:

### I.8.1Computational Fluid Dynamics (CFD) as a methodological foundation:

- CFD is not only a simulation tool but the analytical lens through which airflow, temperature distribution, and ductwork performance are evaluated.
- In this study, CFD provides the means to test multiple duct configurations under desert-specific boundary conditions, ensuring that the results reflect realistic thermal loads.

### I.8.2Thermal Comfort as the human-centered outcome:

- Comfort indices such as PMV and PPD are used as evaluation criteria for the CFD results.

- Rather than being a general definition, thermal comfort here is framed as the measure of success for HVAC design in desert offices, directly linking simulation outputs to occupant well-being and productivity.

**I.8.3Air Conditioning System Design as the engineering variable:**

- Duct number, location, airflow velocity, and outlet temperature are treated as design parameters within the CFD model.
- The framework positions HVAC design as the controllable factor that can be optimized to balance energy efficiency with thermal comfort.

By combining these three dimensions, the framework establishes a cause–effect relationship:

- Design parameters (duct layout, airflow) → influence airflow and temperature distribution (CFD outputs) → which determine thermal comfort indices (PMV/PPD).

This structured approach ensures that the study not only replicates previous CFD applications but also adapts them to the desert office context, where cooling efficiency and occupant comfort are inseparable from energy sustainability.

## I.9Presenting the Scientific Hypotheses Underlying the Research:

Before commencing the simulation and analysis, it is essential to establish the scientific hypotheses that form the foundation of this study. These hypotheses are derived from previous research findings, theoretical principles of thermal comfort, and the unique challenges posed by desert climates. They serve as guiding assumptions that will be tested and validated through numerical simulations.

**Basic assumptions:**

1. **Air distribution within a room depends primarily on duct location and airflow velocity.**

- The positioning of supply and return ducts directly influences airflow patterns, mixing efficiency, and temperature uniformity.
- Variations in duct placement can create zones of stagnation or uneven cooling, while optimized layouts promote balanced air distribution.
- Airflow velocity determines the strength of circulation and the ability to overcome thermal stratification, especially in larger office spaces.

2. **Thermal comfort improves with uniform air distribution and reduced temperature variation.**

    - Occupants experience greater satisfaction when temperature differences across the room are minimized.

- Uniform airflow reduces localized discomfort such as drafts or hot spots.
- Comfort indices (PMV and PPD) are expected to show significant improvement when duct configurations achieve consistent cooling across the occupied zone.

3. **CFD simulation can accurately predict the impact of duct design on indoor climate.**
   - CFD provides a reliable method for modeling airflow velocity fields, temperature gradients, and turbulence effects.
   - By incorporating boundary conditions specific to desert climates, CFD can replicate realistic thermal loads and predict system performance.
   - The hypothesis assumes that CFD results will correlate strongly with theoretical comfort indices, validating its use as a design tool.
4. **Desert climatic conditions amplify the importance of duct location and ventilation design.**
   - High outdoor temperatures and intense solar radiation increase cooling demand, making duct efficiency critical.
   - Low humidity levels reduce the buffering effect of moisture, intensifying the impact of airflow distribution on perceived comfort.
   - Optimized duct placement is hypothesized to significantly reduce cooling loads, thereby improving both comfort and energy efficiency in desert offices.

**Research Application:**

Based on these hypotheses, numerical simulations will be conducted on office rooms located in desert regions, testing various duct locations, airflow velocities, and outlet temperatures. The simulations will evaluate:

- Temperature distribution across the occupied zone.
- Airflow uniformity and circulation efficiency.
- Thermal comfort indices (PMV/PPD) under different configurations.
- Energy efficiency implications of duct design choices.

The outcomes will validate or refute the hypotheses, providing a scientific basis for recommending HVAC design strategies tailored to desert office environments.

## I.10Conclusion:

The review of air-conditioning system performance in hot climates highlights the decisive role of duct location and wall-mounted unit placement in achieving effective cooling and thermal comfort. Numerical studies consistently show that poorly positioned inlets lead to uneven air distribution and thermal stratification, while optimized layouts improve circulation

and reduce discomfort. For example, configurations with multiple ducts lowered room temperatures from approximately 36 °C to 33 °C, while wall-mounted units operating at higher inlet velocities achieved average temperatures around 21 °C to 20 °C, depending on flow rate these quantitative findings confirm that airflow velocity, diffuser arrangement, and inlet positioning are not secondary design choices but primary determinants of both cooling efficiency and occupant satisfaction.

The findings also emphasize the value of Computational Fluid Dynamics (CFD) as a reliable tool for analyzing and predicting air distribution patterns in enclosed spaces. By simulating temperature fields, velocity distributions, and comfort indices such as PMV and PPD, CFD provides insights that support the design of energy-efficient HVAC systems tailored to extreme Saharan conditions the ability to integrate turbulence models, radiation effects, and boundary condition sensitivity analyses makes CFD particularly suited for evaluating HVAC performance in desert climates, where solar loads and ambient temperatures are exceptionally high.

This chapter establishes the theoretical framework and hypotheses for the present study, focusing on how inlet configuration and airflow management can be optimized to ensure thermal comfort and reduce energy demand in desert office environments […. By bridging literature findings with numerical modeling, the study aims to propose validated HVAC strategies that balance occupant well-being with sustainable energy use in hot-arid regions].

# *Chapter II: Methodology of mathematical and numerical formulation*

In this chapter, the numerical simulation is performed using ANSYS Fluent software, based on the Finite Volume Method. The airflow inside of office room an air-conditioned office is considered a 3D, turbulent, incompressible, and steady-state flow. The CFD simulation involves numerically solving the fundamental equations of fluid mechanics and heat transfer.

## II.1 Basic Equations:

### II.1.1 The continuity equation:

The continuity equation expresses the conservation of mass within the computational domainit ensuresthat the net mass flux across the boundaries of a control volume iszero, meaningthat no artificial accumulation or depletion of mass occurs in the numerical model. This principleisfundamental in CFD, as itguaranteesthat the simulatedairflowremainsphysically consistent with real-world behavior. For incompressible flows, wheredensityisassumed constant, the equation simplifies to the divergence of the velocityfieldbeingzero:

$$\nabla \cdot \vec{V} = 0 \quad \text{II.1}$$

This equation means that the mass of air entering a control volume is equal to the mass of air leaving, which is valid for an incompressible flow.

### II.1.2 Mass conservation equation:

The conservation of mass states that the total mass of a fluid remains constant within a control volume. In other words, the mass entering the domain must equal the mass leaving it. This principle is expressed mathematically by the continuity equationit represents the fundamental balance law in fluidmechanics, ensuringthatdensity variations and velocityfields are consistentlyaccounted for in the numerical model. In compressible flows, the full form of the equationisrequired, while in incompressible flowsit simplifies to a divergence-free velocityfield:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{V}) = 0 \quad \text{II.2}$$

This equation ensures that the airflow modeled in the office remains physically consistent, preventing artificial accumulation or loss of mass in the simulation. It is the foundation for analyzing ventilation and air distribution in HVAC systems.

### II.1.3 Conservation of Energy

The energy equation represents the conservation of thermal energy in the fluid. It includes conduction, convection, and internal heat sourcesthis formulation accounts for transientheatstorage, advective transport by airflow, diffusive conduction through the fluid and boundaries, and additionalenergy contributions from occupants, equipment, and lighting. In CFD, solving the energyequationis essential for predictingtemperaturefields and evaluating thermal comfort indices:

$$\rho C_p \frac{\partial T}{\partial t} + \rho C_p(\vec{V} \cdot \nabla T) = k\nabla^2 T + Q \quad \text{II.3}$$

This equation is essential for analyzing temperature variations caused by air conditioning, furniture, and human occupants, directly linking to thermal comfort evaluation.

### II.1.4 TheNavier-Stokes equations:

The Navier-Stokes equations represent the conservation of momentum they describe how velocityfieldsevolveunder the combined influence of pressure gradients, viscous diffusion, and inertial forces. In CFD, theseequations are the cornerstone for predictingairflowbehavior, turbulence, and pressure distribution withinenclosedspaces. Their solution providesdetailed insight into how air-conditioningsystems affect circulation patterns and comfortlevels:

$$\rho(\vec{V} \cdot \nabla)\vec{V} = -\nabla P + \mu\nabla 2\vec{V} \quad \text{II.4}$$

These equations allow us to determine the distribution of air velocity and pressure in the air-conditioned room. Where:ρ : Air density ($kg/m^3$), $\vec{V}$ : velocity vector ($m/s$), $P$ : Pressure ($Pa$), $\mu$ : Dynamic viscosity ($kg/m{\cdot}s$).

### II.1.5 The energy equation:

The energy equation describes heat transfer within the room:

$$\rho c_P(\vec{V} \cdot \nabla T) = k\nabla^2 T \quad \text{II.5}$$

Where$T$: Temperature($K$), $C_P$: Specific heat ($J/kg{\cdot}K$), $k$ : Thermal conductivity ($W/mK$).

This equation allows us to calculate the temperature distribution within the office room based on convection and thermal conduction. These equations are discretized and solved numerically on each cell of the mesh.

### II.1.6 Turbulence Model:

Airflow in an air-conditioned of office room is generally turbulent due to air velocities and obstacles (walls, furniture, occupants).

To model this type of flow, a turbulence model must be used.In this study, the standard $k-\varepsilon$ model is used because it is widely employed in CFD simulations of internal flows and offers a good compromise between accuracy and computation timeThe k–ε model isparticularly effective for predicting recirculation zones, mixinglayers, and jet diffusion, which are common in HVAC applications. Itsrobustness and relativelylowcomputationalcostmakeitsuitable for large-scale simulations such as office environments, wheredetailed turbulence resolution (e.g., LES or DNS) wouldbeprohibitivelyexpensive.

The $k-\varepsilon$ model is based on solving two transport equations: Turbulent kinetic energy equation $k$. Turbulent dissipation rate equation $\varepsilon$. These equations allow us to model the effect of turbulence on the flow and heat transfer.

The $k-\varepsilon$ model is particularly well-suited for:Internal flows**,** Ventilation and air conditioning**,** Flows with air recirculation.

## II.2 Thermal Comfort:

Thermal comfort is defined as an individual's level of satisfaction with the thermal environment. It depends on several physical and physiological parameters. In this study, thermal comfort is assessed based on air temperature, air velocity and temperature distribution within the room. The most commonly used thermal comfort indices are:

### II.2.1 Predicted Mean Vote (PMV):

An index that predicts the average perceived temperature of occupants on a scale of -3 to +3Developed by Fanger, the PMV model integratesenvironmentalparameters (air temperature, mean radiant temperature, humidity, and air velocity) withpersonalfactors (metabolic rate and clothinginsulation). It provides a quantitative measure of thermal sensation, linkingphysical conditions to humancomfort perception**.**

$$PMV = [0.303 \cdot e^{-0.036M} + 0.028] \cdot \{(M - W) - 3.05 \cdot [5.73 - 0.007(M - W) - (M - W)] - 0.42 \cdot [(M - W) - 58.15] - 0.0173M(5.87 - P_a) - \quad \textbf{II.6}$$

Where:

- $M$: Metabolic rate (W/m²)
- $W$: External work (W/m²)
- $T_a$: Air temperature (°C)
- $T_r$: Mean radiant temperature (°C)
- $P_a$: Water vapor partial pressure (Pa)
- $f_{cl}$: Clothing surface area factor
- $T_{cl}$: Clothing surface temperature (°C)
- $h_c$: Convective heat transfer coefficient

**Table II.1** Thermal sensation scale according to the PMV index.

| Thermal Sensation | PMV |
|---|---|
| Very Cold | -3 |
| Cool | -2 |
| Slightly Cool | -1 |
| Neutral | 0 |
| Slightly Warm | +1 |
| Warm | +2 |
| Very Warm | +3 |

### II.2.2 Predicted Percentage of Dissatisfied (PPD):

An index that estimates the percentage of people who are thermally dissatisfied. For good thermal comfort, PMV should be between -0.5 and +0.5, and PPD should be less than 10%.

$$PPD = 100 - 95 \cdot e^{-0.03353 \cdot PMV^4 - 0.2179 \cdot PMV^2} \quad \textbf{II.7}$$

## II.3Control Domain (Geometry):

The computational domain represents an air-conditionedoffice modeled in three dimensions using ANSYS DesignModeler software.

The room dimensions of$3.6m \times 4.1m \times 3.1m$.

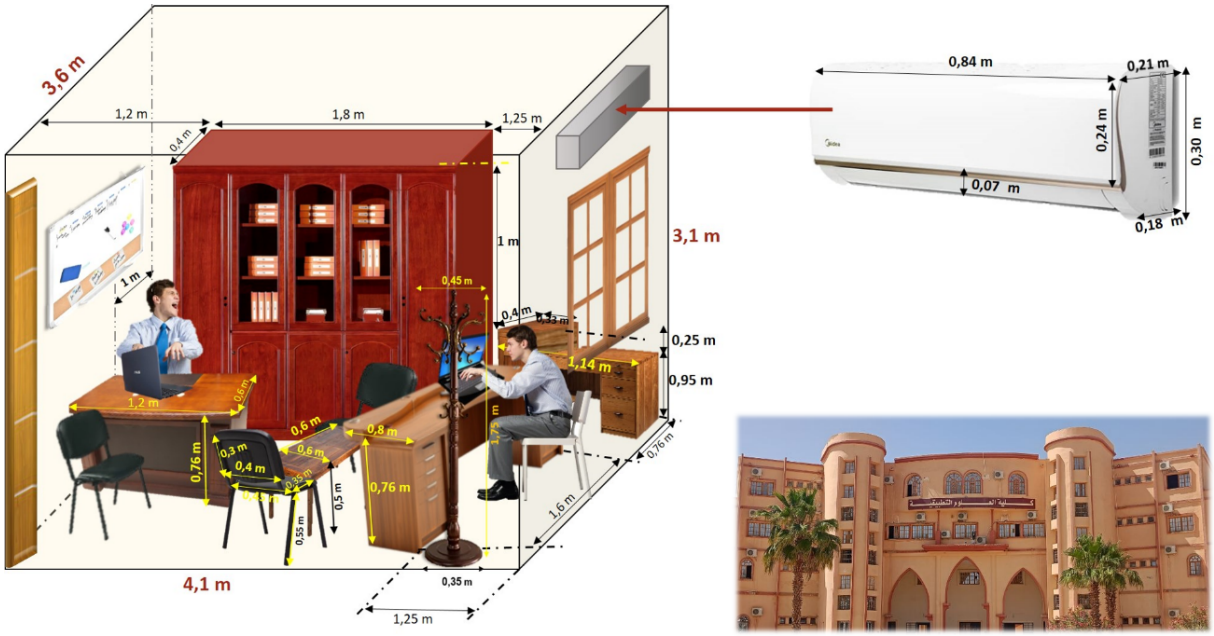



**Figure II.1:**Location, components and dimensions of the office at Applied Sciences Faculty.

To enhance airflow distribution and reduce stratification, a new wing design was developed in ANSYS DesignModeler. The office geometry was modified to include multiple supply openings positioned at different levels, ensuring that conditioned air could reach both upper and lower regions of the occupied zone.The design process involved creating extrusions to represent the inlets and applying Boolean operations to integrate them into the main channel. This arrangement was selected to promote balanced circulation, improve mixing, and provide a more uniform cooling effect compared to the single-vent configuration.

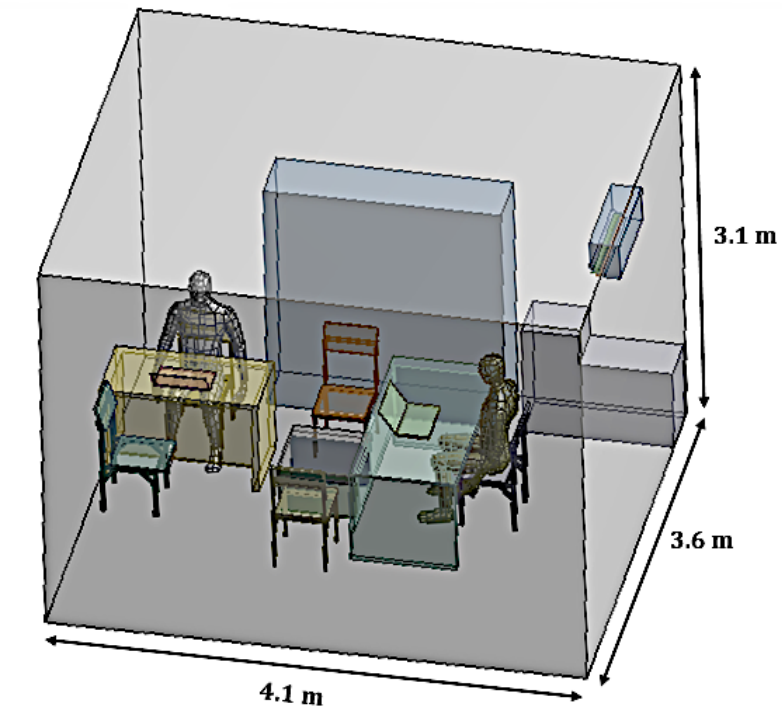



**Figure II.2:** 3D view of the office with ANSYS Design.

The resulting geometry served as the basis for Case 2 simulations, enabling direct comparison with the normal inlet case and demonstrating the potential of distributed supply strategies in Saharan office environments.

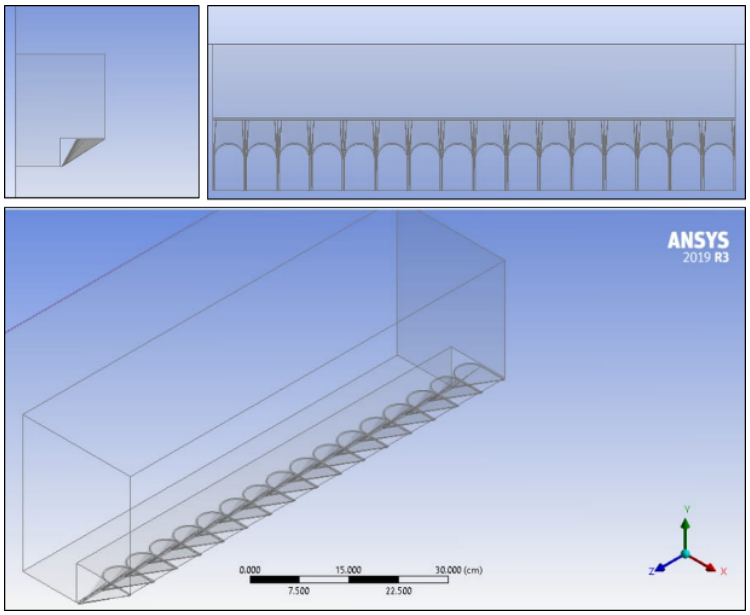



**Figure II.3:** Form of new wing designed in ANSYS Design Modeler.

## II.4Boundary Conditions and Materials:

Boundary conditions were defined to represent realistic operating conditions of the air conditioned office, conditions presented in Table II.2.

**Table II.2:**Applied experimental boundary conditions (Values reflectSaharan office conditions for CFD comfortanalysis).

| | |
|---|---|
| Supply air velocity: | 1.8 m/s (constant for all cases). |
| Supply air temperature: | 21.1 °C |
| Recommended indoor temperature: | 25 °C |
| Indoor walls mean temperature: | 26 °C |
| Outdoor walls mean temperature: | 40 °C |
| Relative humidity: | 34.5 % |
| Furniture temperature: | 24 °C |
| Occupants temperature: | 37 °C |
| Computer temperature: | 28 °C |

## II.5Mesh study:

For the computational domain, the mixt mesh was generated to discretize the geometry and enable numerical solution of the governing equations of fluid flow and heat transfer. The choice of tetrahedral elements was motivated by their flexibility in handling complex geometries, particularly those involving irregular boundaries, curved surfaces, and internal obstacles such as furniture and human occupants. This type of mesh allows for accurate representation of the office environment while maintaining computational efficiency.

To improve solution accuracy, local mesh refinement was applied in critical regions of the domain. These included air supply inlets and outlets, where velocity gradients are high and turbulence effects are significant.Near-wall regions, where boundary layer development influences heat transfer and airflow distribution.Around obstacles such as furniture and occupants, where flow separation and recirculation zones are expected.

The final mesh provided a balance between resolution and computational cost. While refinement was concentrated in regions of high flow activity, coarser elements were used in less critical zones to reduce the overall cell count and maintain feasible simulation times. This adaptive meshing strategy ensured that the model captured the essential physics of airflow and temperature distribution without excessive computational demand.

### II.5.1 Mesh convergence:

A grid independence test was carried out to ensure that the numerical results were not dependent on mesh resolution. Three mesh levels coarse, medium, and fine were generated and compared in terms of velocity distributions within the computational domain. The results showed negligible differences between the medium and fine meshes, indicating that further refinement did not significantly improve accuracy. Therefore, the medium mesh was selected for the simulations, as it provided a good balance between computational accuracy and efficiency, minimizing simulation time while maintaining reliable results.

It is important to highlight that enhanced case (configuration with new wing design) required a finer mesh compared to Case 1. The presence of two inlets—one located at the upper level and another at the lower level—introduced more complex airflow structures within the computational domain. These included recirculation zones, vertical stratification, and stronger interaction between supply streams, all of which demanded higher spatial resolution to be captured accurately.

To address this, additional mesh refinement was applied in the regions surrounding both inlets and along the zones where airflow interaction occurred. This ensured that velocity gradients, turbulence effects, and temperature variations were properly resolved, thereby improving the reliability of the simulation results for Case 2.

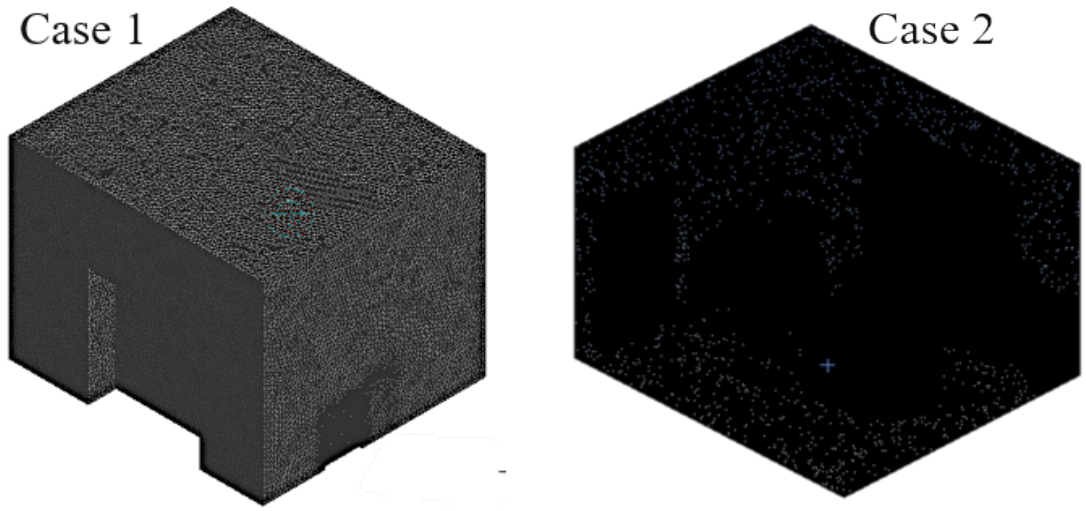



**Figure II.4:** Mesh distribution for Case 1 (with simple wing) and Case 2 (with enhanced wing).

### II.5.2Solution convergence:

The convergence of the numerical solution was assessed using several indicators as: Residuals: The iterative solver tracks residuals of continuity, momentum, energy, and turbulence equations. Convergence was considered achieved when residuals dropped below $10^{-3}$ for continuity, momentum, and turbulence equations, and below $10^{-6}$ for the energy equation.Stability of monitored variables: Average air temperature, velocity magnitude at selected points, and heat transfer rates were monitored during iterations. Convergence was confirmed when these values stabilized and showed negligible variation between successive iterations.Energy consistency: Since the domain has no outlet, conservation was checked by ensuring that the total heat added (from walls, furniture, and human bodies) balanced with the cooling effect of the inlet air. This confirmed that the energy equation was properly satisfied.

This multi-criteria approach ensured that the solution was not only numerically converged but also physically consistent.

## II.6Numerical solution validation:

Validation was performed to ensure the accuracy, reliability, and physical consistency of the CFD model before proceeding with detailed case analysis. Several complementary approaches were applied to confirm that the numerical methodology was robust and suitable for studying airflow and cooling performance in Saharan office environments.

### II.6.1 Comparison with experimental results:

The numerical results were compared against our own real experimental results on indoor airflow and cooling performance of the office under hot arid conditions at Ouargla University (Applied Sciences faculty). This comparison provided perfect validation and confirmed that the simulation approach was consistent with experimental findings.

### II.6.2 Physical plausibility:

The simulation reproduced expected physical behavior observed in real office environments. Cooler air was concentrated near the supply diffusers, warmer zones formed farther away from the inlets, and vertical stratification effects were evident in regions of reduced circulation. These results aligned with theoretical expectations of buoyancy-driven flow and convective heat transfer, reinforcing confidence in the model's ability to capture realistic airflow dynamics.

### II.6.3 Mesh independence:

A **meshindependenceanalysis**wasperformed to ensurethat the numericalresultswere not overlydependent on gridrefinementbeyond the medium mesh. Velocity and temperature profiles werecomparedacrosscoarse, medium, and fine grids, and only marginal differenceswereobservedbetween the medium and fine resolutions. This confirmedthat the selectedmeshdensityprovidedadequateaccuracywhilepreservingcomputationalefficiency, guaranteeingthatnumericalartifactsdid not distort the physicalinterpretation of the results.

### II.6.4 Boundary condition consistency:

The imposed thermal boundary conditions including wall materials, furniture, and human body heat sources produced realistic gradients and airflow responses. The interaction between supply air and internal heat loads was consistent with expected thermal behavior, validating the assumptions used in defining boundary conditions and material properties.

## II.7 Conclusion

This chapter presented the theoretical and numerical foundations necessary for CFD simulation applied to thermal comfort in Saharan climates of Ouargla. The conservation equations for mass, momentum, and energy form the physical basis of the model, while the integration of a turbulence model (standard k ε) allows for a realistic representation of air recirculation and stratification phenomena. The convergence study and numerical validation confirmed that the solutions obtained are stable, consistent, and conform to international thermal comfort standards.Thus, this part establishes the essential methodological foundation for the analysis of the various cases studied of wings configurations of indoor cooling unit.

# *Chapter III: Results and Discussions*

## II.1Introduction

This chapter presents the numerical results obtained from the CFD simulations of the air conditioningforoffice environment. The simulations were designed to evaluate airflow behavior, temperature distribution, and thermal comfort under different outlet wing designs of the indoor cooling unit configurations. Two distinct cases were analyzed to examine the influence of wing design on cooling performance:

The first case is a simple wing of the indoor cooling unit. In this configuration, conditioned air was supplied through a single outlet, representing a conventional conditioning design. The analysis focused on how airflow from a simple outlet distributes across the office space, its ability to maintain uniform cooling, and the resulting comfort levels for occupants.The second case is an enhanced outlet wing equipped with equipped with fins. This configuration is designed to investigate how the outlet affects creating more complex airflow structures such as recirculation zones and vertical stratification, and whether this configuration improves cooling efficiency and occupant thermal comfort compared to the simple wing case.

The results are discussed in terms of temperature distribution, by identifying hot and cold office parts and stratification effects, and checking the uniformity of cooling. Likewise, the velocity fields examining airflow strength, also the circulation patterns, and potential draft discomfort. Also showing the airflow streamlines visualizing flow paths to understand mixing efficiency and circulation behavior. Finally, deduce the impact on thermal comfort from evaluating comfort indices PMV and PPD to quantify occupant satisfaction under each configuration.

This study aims to determine the optimal outlet of the indoor cooling unit for office environments under hot arid conditions of Ouargla and balancing energy efficiency with thermal comfort. The following sections present detailed temperature and velocity contours, streamlined visualizations, and quantitative analyses that form the basis for evaluating the performance of each configuration.

## III.2 Validation of the Numerical Model:

Validation is an essential step to ensure the reliability of the results obtained from the CFD simulation. It consists of comparing the numerical data obtained with experimental results this process establishes confidence that the computational model reproduces physical reality within acceptable error margins.

Comparison of experimental measurements of ambient air temperatures with the simulated results in the office environment showed that the average relative error remained below 5%, which confirms the accuracy of the model.

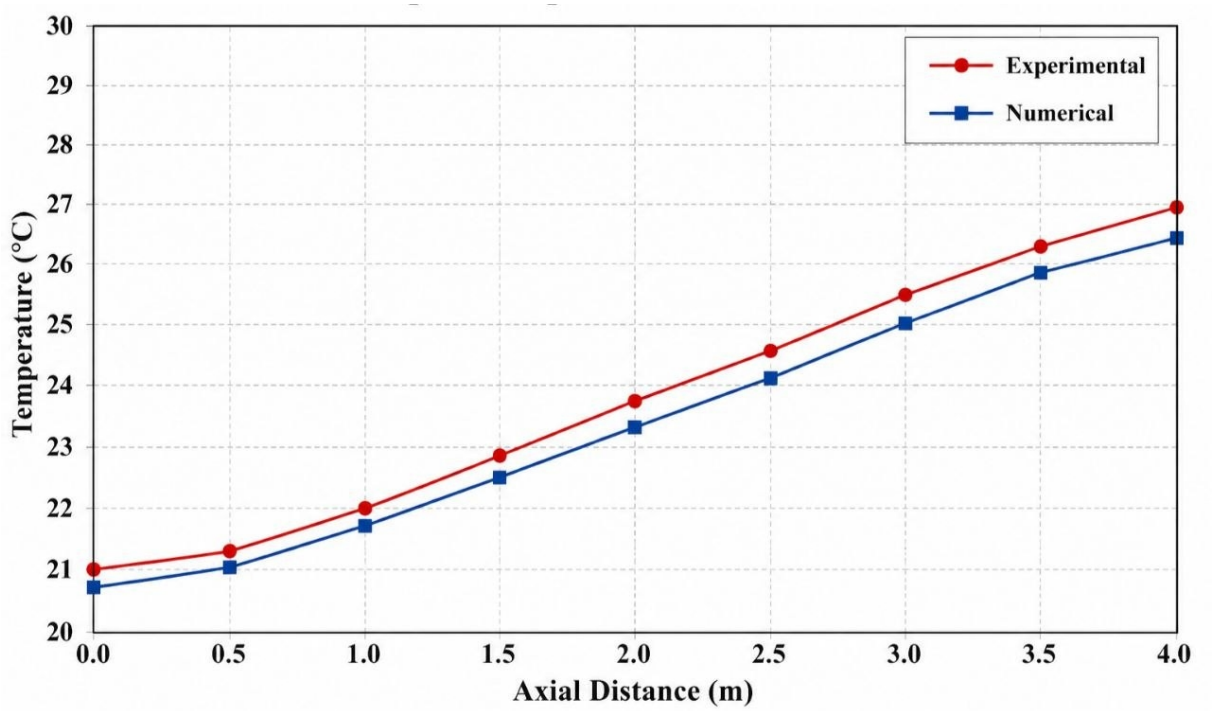



**Figure III.1 Comparison of experimental and numerical axial temperature profiles of the tested cases.**

## III.3Case 1: Simple wing:

Cooling performance in this configuration was concentrated in the immediate vicinity of the supply vent, where conditioned air was out at a lower temperature of 21.1 °C and a higher velocity of 1.8 m/s. This produced a localized comfort zone near the inlet, characterized by strong mixing and noticeable cooling. However, regions farther from the vent remained significantly warmer, with maximum temperatures reaching 27 °C. Average velocities in the occupied zone dropped to 0.25 m/s, confirming infirm circulation and limited airflow crosses.These warmer parts highlight the limitations of simple wing design, where air distribution is uneven and pronounced temperature gradients develop across the office.

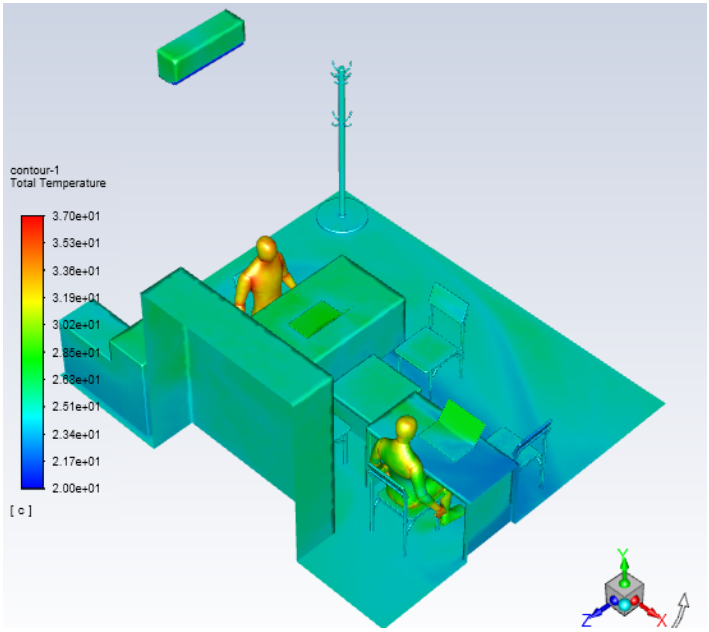



**Figure III.2:** 3D temperature distribution in Case 1.

A distinct vertical stratification was also evident. Cooler air remained concentrated near the outlet and at lower levels, while warmer air accumulated in upper or remote areas due to buoyancy effects and insufficient mixing. This behavior reflects the natural tendency of warm air to rise and cooler air to settle, producing uneven thermal conditions.

From a comfort perspective, PMV values ranged from −0.75 near the outlet to +0.9 in warmer regions, while PPD exceeded 12%, indicating that a significant proportion of occupants would feel dissatisfied.Shortly, the case of the simple wing of the indoor cooling unit demonstrated that it can provide localized cooling; it suffers from poor distribution, thermal stratification, and reduced comfort for occupants positioned farther from the supply source.

These findings emphasize the importance of the outlet design for airflow in achieving more uniform cooling, minimizing discomfort, and improving energy efficiency in office environments.

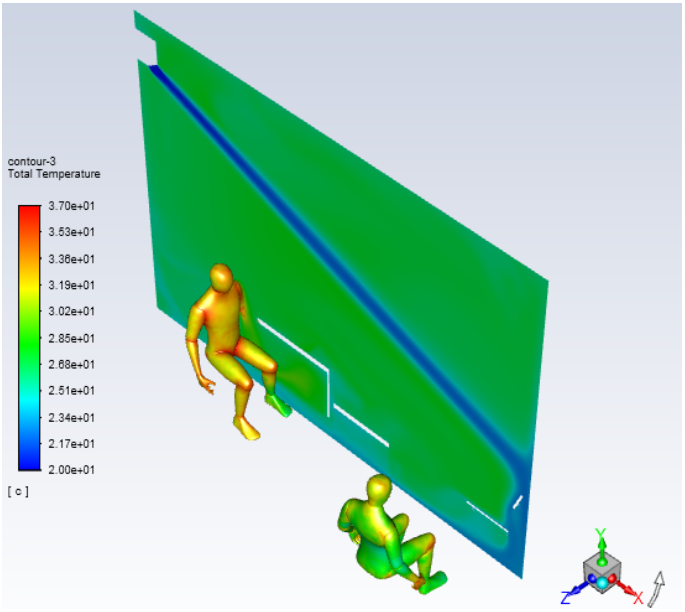



**Figure III.3:**Temperature contours on center vertical plane in Case 1.

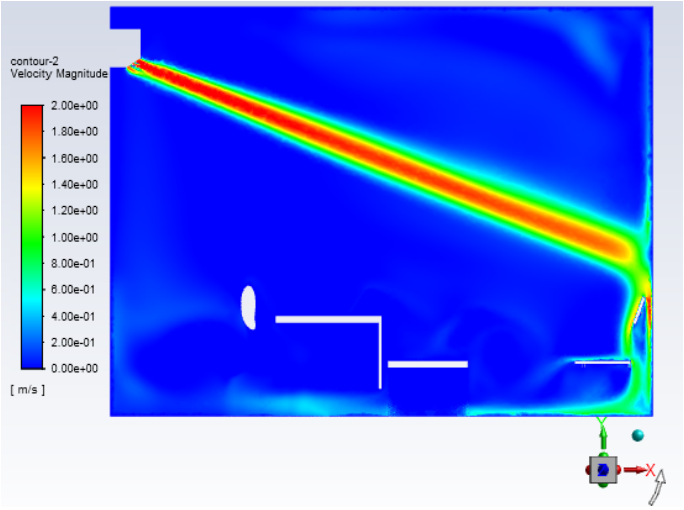
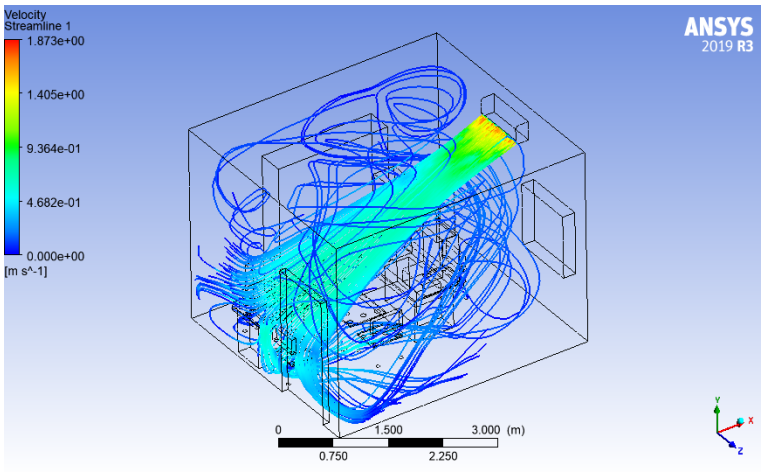



**Figure III.4:** Velocity on center vertical plane in Case 1.

The streamline plots visualize airflow paths within the office, confirming that circulation is strongest near the supply outlet but diminishes as the flow penetrates deeper into the domain. Close to the inlet, the streamlines are dense and well-defined, reflecting strong momentum and effective mixing.

However, farther from the vent, the streamlines become sparse and irregular, indicating weaker circulation and reduced coverage. This uneven distribution highlights the limited ability simple wing to sustain airflow throughout the entire space, leaving certain regions under ventilated and prone to thermal discomfort.




**FigureIII.5:** Streamlines of Case 1.

The analysis of thermal comfort indices confirmed that cooling within the office was uneven. Occupants located near the supply vent benefited from cooler air and stronger circulation, with local temperatures around 20 to 22 °C and air velocities up to 2 m/s. In these regions, PMV values remained close to neutral −0.25 to 0.45, while PPD percentages stayed below 8%, indicating that most individuals would feel thermally satisfied.

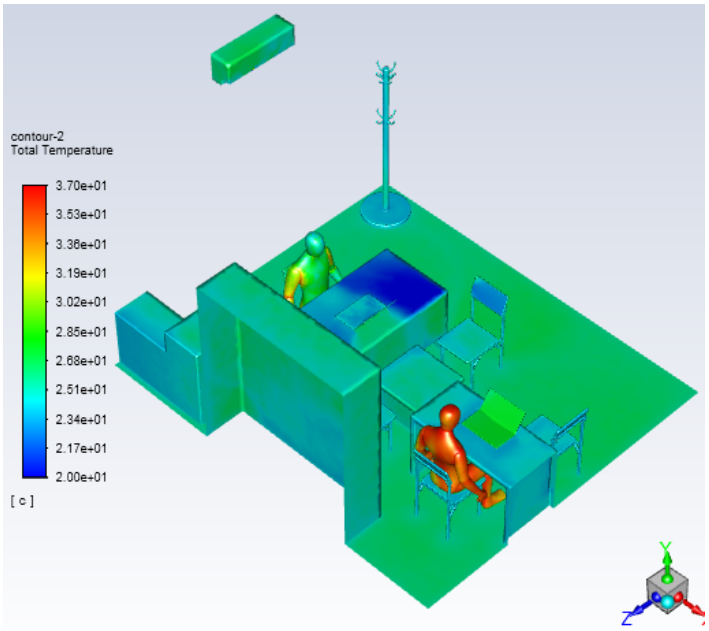
This localized comfort zone reflected the direct influence of conditioned air delivery and highlighted the effectiveness of cooling in the immediate vicinity of the inlet.

In contrast, occupants positioned farther from the vent experienced significantly higher temperatures, with values rising to 27-29 °C and velocities dropping to 0.2 to 0.25 m/s. Comfort indices in these regions deteriorated, with PMV values increasing to +0.75 to +0.9 and PPD exceeding 12-15%, suggesting warmer sensations and greater dissatisfaction. The reduced airflow penetration and presence of stagnant zones contributed to weaker cooling performance, reinforcing the uneven distribution of comfort across the domain.

Vertical stratification further amplified this effect. Temperature differences between lower and upper levels exceeded 9 °C, leaving occupants at higher elevations more exposed to discomfort. Overall, the findings emphasized that while the system provided adequate comfort near the vent, its performance was limited by poor mixing and uneven coverage.

## III.4Case 2: Enhanced wing

For this developed case, the wing of the indoor cooling unit provided by fins shows a horizontal coverage that was notably improved, ensuring that occupants positioned farther from the outlet cooling unit experienced more consistent cooling. The temperature difference between the inlet zone and remote regions narrowed to less than 4 °C, compared to differences exceeding 9 °C in the simple wing.



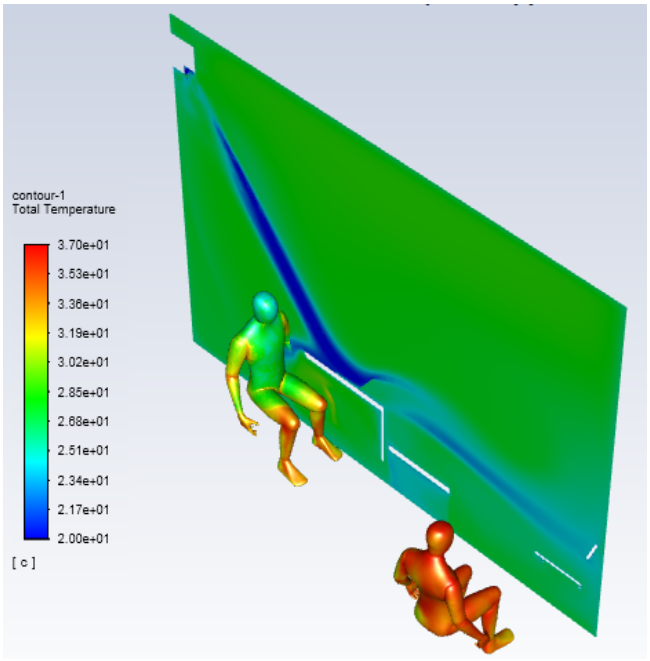

**FigureIII.6:** 3D temperature distribution in Case 2.

The conditioned air outleted through enhanced wing at 21.1 °C, with inlet velocities of approximately 1.8 m/s. This new desing enhanced circulation, allowing cold air to cross deeper into the room compared to simple wing case. As a result, localized hot spots were significantly reduced, with maximum temperatures limited to 23 °C instead of 27 °C observed in Case 1. Average velocities in the occupied zone increased to 0.63 m/s, confirming improved airflow penetration and mixing.

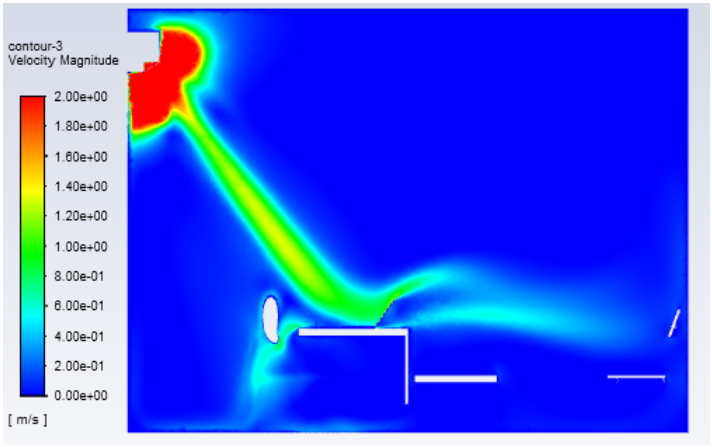
Vertical stratification was also minimized. Cooler air was delivered at both upper and lower levels, disrupting the layering effect observed previously in case 1. Warm air accumulation in upper regions was eliminated, producing a smoother thermal field. Comfort indices reflected this improvement: PMV values ranged from −0.35 to +0.45, while PPD dropped below 6%, indicating that the majority of occupants would feel thermally satisfied.

Basically, the case 2 demonstrated perfect cooling performance, with balanced temperature distribution, reduced stratification, and agreement of thermal comfort indices with standards. These findings confirm that the distributed outlet of enhanced wing design

offers both energy efficiency and improved occupant comfort, making it a more effective solution for Saharan office environments compared to the conventional wing design.




**FigureIII.7:** Temperature contours on center vertical plane in Case 2.



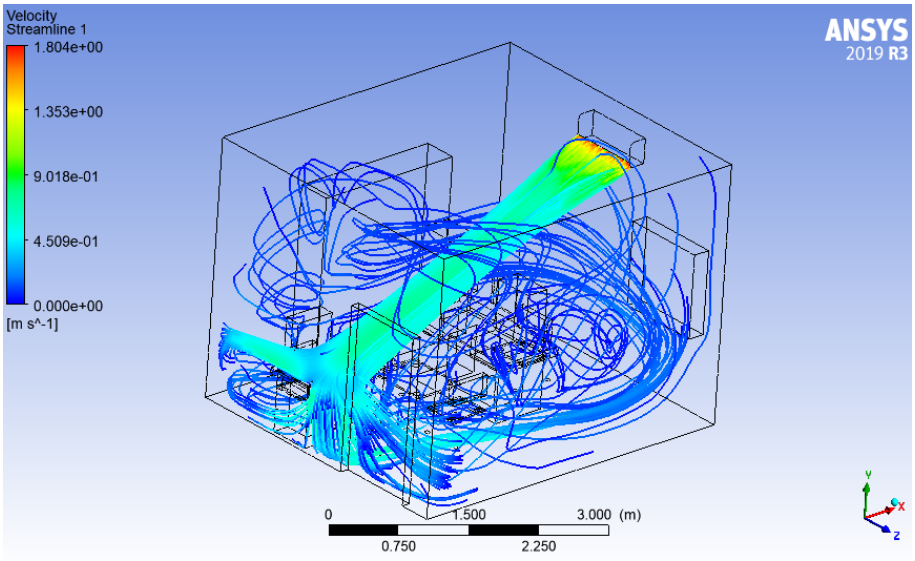

**FigureIII.8:** Velocity distribution on center vertical plane in Case 2.

The velocity field results clearly displayed improved circulation when enhanced wing was employed. It showed strong jets of conditioned air outlasted; unlike Case 1, where velocity decayed rapidly, the new design arrangement sustained airflow momentum across a larger area, with average velocities in the occupied zone reaching 0.63 m/s compared to 0.25 m/s in the simple wing case. As a result, stagnant pockets were minimized, and the overall velocity distribution appeared more uniform, confirming that the new enhanced design provided superior coverage of the occupied zone.

Consequently, the velocity fields demonstrated that the new wing design not only enhanced circulation but also supported more consistent cooling and improved thermal comfort throughout the office environment.

The streamlined preset in the figure III.9 below clearly illustrates enhanced mixing when the outlet with an enhanced wing is showing that airflow paths are more evenly distributed across the office domain.

Compared to the outlet with conventional wing, the presence of fins creates more strength in circulation and reduces the decay of momentum as the flow penetrates deeper into the room. This improved distribution minimizes stagnant pockets and ensures that conditioned air reaches more areas, confirming that the new wing design provides superior coverage of the occupied zone. The denser and more interconnected streamlines highlight the effectiveness of the configuration in sustaining airflow throughout the office space.




**FigureIII.9:** Streamlines of Case 2.

## III.5Comparative Discussion:

**Table 9**Comparative results of study cases

| Aspect | Case 1: Simple wing | Case 2: Enhanced wing |
|---|---|---|
| Temperature uniformity | Cooling concentrated near vent; 20–23 °C near inlet, but hot spots up to ~27 °C farther away; vertical differences exceed 09 °C | More uniform cooling; cold air penetrates deeper; maximum temperatures limited to ~23 °C; vertical differences reduced to <5 °C |
| Velocity distribution | Strong airflow near vent (~2.0 m/s); average velocities drop to ~0.4 m/s; stagnant zones with <0.2 m/s | Improved circulation; dual jets at 1.8 m/s; average velocities increase to ~0.6 m/s; stagnant regions minimized |
| Streamlines | Limited mixing; airflow follows direct path; poor coverage of domain | Enhanced mixing; overlapping flow zones; better coverage of entire office |
| Thermal comfort (PMV/PPD) | Uneven comfort; PMV ranges from −0.75 near vent to +0.9 in warm zones; PPD exceeds 12% | Improved comfort; larger percentage of occupants within acceptable range; lower dissatisfactionPPDreach ~5% |

Generally, the comparison highlights the importance of the outlet on indoor cooling unit configuration in conditioning design (Table III1). From Case 1 (simple wing), it provides basic cooling but suffers from poor distribution and thermal stratification. Case 2 (enhanced wing) demonstrates superior performance, with more uniform temperature, stronger circulation, and improved thermal comfort indices according to international standards.

## III.6 Conclusion:

The results confirm that while a simple wing provide localized cooling, it is insufficient for achieving uniform comfort in a Saharan office environment. In the case of simple wing, conditioned air entered at 21.1 °C with a velocity of 1.8 m/s, producing a comfort zone near the vent. However, cooling performance diminished with distance, leaving hot spots up to 27 °C and stagnant regions with velocities below 0.2 m/s. Vertical stratification was pronounced, with temperature differences exceeding 9 °C between lower and upper levels. Comfort indices reflected this imbalance: PMV ranged from −0.75 near the vent to +0.9 in warmer regions, while PPD exceeded 12–15%, confirming significant occupant dissatisfaction.

In contrast, the new wing design provided superior cooling performance. Enhanced form supplied air at 20 °C with inlet velocities of 1.8 m/s, enhancing circulation and mixing. Maximum temperatures were limited to 23 °C, while average velocities in the occupied zone increased to 0.6 m/s, reducing stagnant pockets. Vertical stratification was minimized, with temperature differences reduced to less than 5 °C. Thermal comfort indices improved markedly: PMV values ranged from −0.35 to +0.45, and PPD dropped below 5%, indicating that the majority of occupants experienced thermal satisfaction.

Finally, the comparative analysis demonstrated that the distributed design of the new wing ensures better airflow distribution, reduced stratification, and improved thermal comfort. By narrowing temperature gradients and lowering dissatisfaction rates, this configuration offers both energy efficiency and greater occupant satisfaction, making it a more effective solution for Saharan office environments than the conventional design.

## General Conclusion:

This study has addressed the challenge of achieving thermal comfort in Saharan office environments on Ouargla, where extreme climatic conditions make cooling both essential and difficult.

The present work is established by highlighting the importance of air conditioning systems in hot arid regions and the need for efficient airflow distribution to ensure thermal comfort for occupants. The findings yielded emphasized the role of Computational Fluid Dynamics (CFD) as a reliable tool for analyzing indoor environments of offices, providing the foundation for the methodology adopted in this work.

The mathematical and numerical formulation was presented in detail, and governing equations of fluid flow and heat transfer were introduced, along with the agreed turbulence model of standard k-ε selected to capture the indoor airflow.

Thermal comfort indices such as PMV and PPD are used to determine and evaluate system effectiveness. Moreover, mathematical and numerical steps were presented in detail, and governing equations of fluid flow and heat transfer were introduced. Along with the turbulence model of standard k-ε selected to capture the complexity of indoor airflow. The office geometry was modeled with real dimensions and boundary conditions of the hot arid Saharan region of Ouargla.

Also, including materials, furniture, and human occupants. Mesh quality and convergence were carefully studied to ensure numerical accuracy, and the solution was validated relative to the experimental values, confirming the reliability of the chosen approach.

The results of different configurations of outlet wing for an indoor cooling unit show that simple wing cooling was concentrated near the vent, with conditioned air entering at 21.1 °C and 1.8 m/s. Localized thermal comfort zones formed, but hot spots persisted with temperatures up to 27 °C and stagnant regions where velocity dropped below 0.2 m/s.

Vertical stratification was pronounced, with temperature differences exceeding 09 °C between lower and upper levels. Thermal comfort indices reflected this imbalance: PMV ranged from −0.75 near the vent to +0.9 in warmer regions, while PPD exceeded 12–15%, confirming significant occupant dissatisfaction.

On the other hand, an enhanced wing with a distributed outlet supplied air at 20 °C with inlet velocities of 1.8 m/s, enhancing circulation and mixing. Maximum temperatures were limited to 23 °C, while average velocities in the occupied zone increased to 0.6 m/s, reducing stagnant pockets.

Vertical stratification was minimized, with temperature differences reduced to lower than 5 °C. Further, thermal comfort indices improved sufficiently, PMV values ranged from −0.35 to +0.45, and PPD dropped below 5%, indicating that the majority of occupants experienced thermal satisfaction.

The findings confirmed that the outlet wing design of the indoor cooling unit plays a decisive role in cooling and ventilation performance (Conditioning system). So from that recorded results with enhanced wing offering perfect cooling allow to reduce stratification and improve the indoor thermal comfort. That means an efficient conditioning process.

This work has demonstrated that CFD simulations are a powerful tool for evaluating and optimizing air-conditioning systems in the Saharan zone. By systematically analyzing airflow and temperature distribution, it was possible to identify weaknesses in conventional designs and propose improved configurations.

The findings contribute to the broader effort of designing energy-efficient and comfortable indoor environments in regions where cooling demand is critical as the hot aide zone of Ouargla, with numerical evidence showing that the enhanced wing configuration of the indoor cooling unit outperform the simple wing in both comfort and efficiency during the daily office cooling process.

# Perspectives:

This study is only a first step in advancing the outlet wing design of the indoor conditioning unit. We can list the following perspectives for thistopic:

1- Explore more refined wing geometry such as different shapes, angles,orwing fins arrangements.
2- Proposed several outlet designs, as separated the outlet from manymoving parts and direction for further air recirculation.
3- Extend the indoor air analysis for more carpets quality, which willprovide a more holistic evaluation (humidity, $CO_2$, Lighting, ….,).

# *References*

[1] ANSYS Fluent Theory Guide. (2024). ANSYS Inc.

[2] Patel, A., &Dhakar, P. S. (2018). CFD Analysis of Air Conditioning in Room Using ANSYS Fluent. Journal of Emerging Technologies and Innovative Research (JETIR), November 2018, Volume 5, Issue 11, pp. 436–441.

[3] Liu, Y., et al. (2021). Optimization of air supply parameters for stratum ventilation from the perspective of controlling airborne diseases and thermal comfort.

[4] Zhu, H., Hu, S., Wang, G., Han, L., Jing, M., & Zhao, X. (2023). Full-scale numerical simulation and experimental validation of the thermal environment under heating mode in an air-conditioned room. Case Studies in Thermal Engineering, 51, 103548.

[5] Ozsagiroglu, S., Camci, M., Taner, T., Acikgoz, O., Dalkilic, A. S., &Wongwises, S. (2021). CFD analyses on the thermal comfort conditions of a cooled room: a case study. Journal of Thermal Analysis and Calorimetry, 147, 2615–2639.

[6]Fanger, P. O. (1970). Analysis and Applications in Environmental Engineering. McGraw-Hill Book Company.

[7]ASHRAE Standard 55-2020. Thermal Environmental Conditions for Human Occupancy. American Society of Heating, Refrigerating and Air-Conditioning Engineers.

[8] ISO 7730:2005. Ergonomics of the Thermal Environment – Analytical Determination and Interpretation of Thermal Comfort Using Calculation of the PMV and PPD Indices and Local Thermal Comfort Criteria. International Organization for Standardization.

[9] Versteeg, H. K., &Malalasekera, W. (2007). An Introduction to Computational Fluid Dynamics: The Finite Volume Method. 2nd ed. Pearson Education.

[10] Incropera, F. P., DeWitt, D. P., Bergman, T. L., &Lavine, A. S. (2011). Fundamentals of Heat and Mass Transfer. 7th ed. Wiley.

[11] Awbi, H. B. (2003). Ventilation of Buildings. 2nd ed. Department of Construction Management and EngineeringUniversityof.ReadingSpon Press.

[12] Zhai, Z. J., & Chen, Q. (2005). Performance of coupled building energy and CFD simulations. Energy and Buildings, 37(4), 333–344.